F.Rohner

December 18, 1989
PS/PA/Note/89-31

ANSYS
Revision 4.4 Tutorial Files

Introduction

This note contains the preprocessing- (e.g.PREP7) and/or postprocessing- (e.g.POST1) files used in the examples presented in the tutorials for the revision 4.4 of the ANSYS finite element package. All files are extracted from a floppy disk delivered by the SWANSON Company Houston USA.

The files are ordered into the directories listed below. For the list of files contained in these directories see APPENDIX A at the end of this note.

TUTORIAL Number	DIRECTORY on floppy	TUTORIAL TITLE
DN-T021	COUPL_FD	COUPLED FIELD ANALYSIS
DN-T023	R44_CHGS	REVISION 4.4 CHANGES TO THE ANSYS PROGRAM
DN-T018	POSTPROC	POSTPROCESSING
DN-T016	NONL_DYN	NONLINEAR DYNAMICS
DN-T017	STIF43	THE FOUR-NODE PLASTIC SHELL ELEMENT
DN-T022	SOLID	SOLID MODELING
DN-T020	COMPOSIT	COMPOSITE STRUCTURES
DN-T019	FRACTURE	FRACTURE MECHANICS USING THE ANSYS PROGRAM
DN-T015	ACS_WAVE	ACOUSTIC PRESSURE WAVE PROPAGATION

The example files should be used in close conjunction with the manual for each tutorial. Be aware that the files, or the graphics and results produced from them, may differ slightly from those published in the tutorials. It may be necessary to change the arguments on the /SHOW commands in these files to the appropriate form to accommodate your system. Some directories may contain documentation files listed as DOCU1,DOCU2, etc.

ATTENTION : one file is missing here . The file for the SHUTTLE TURBINE BLADE is not contained in this note due to its large size of over 3000 records. This file must be copied separately either from the floppy or from VMARCHIVE.

HOW TO GET THE FILES : This note is stored on CERNDOC : project MAGNETICS ; category REV44_ANSYS ; filename TU44FILE SGML * A copy can be obtained on the A disk via the DOCPRINT panel of CERNDOC (option COPY).

Contents

Introduction i

Chapter 1: Coupled Field Analysis : Tutorial DN – T021 1

1.1 Tokamak Fusion Reactor Coil Analysis STIF 5 1
1.2 Hot Die Forging Thermal – Stress Analysis (STIF 13, STIF 34) 3
1.3 Slot Embedded Conductor (STIF 13) Eddy Currents, Skin Effect 6
1.4 Double – Mass Loaded Transducer STIF 5 8

Chapter 2: Revision 4.4 Changes Tutorial 11

2.1 Time Step Optimisation STIF 42 11
2.2 Spin Softening STIF 45 13
2.3 Example to Compute Heat Loss Through Sides STIFF 55 14
2.4 Slot Embedded Conductor STIF 13 15
2.5 Axisymmetric Flow – Flow between Coaxial Cylinders STIF 15 16
2.6 Thermal Effects in HAGEN – POISEUILLE Flow STIF 15 17
2.7 Test STIF 43 Elastically STIF 43 18
2.8 Optimisation of Fibre Angle of Composite Material STIF 46 19
2.9 Test STIF 26 with three Nodes and Large Rotation STIF 26 21
2.10 Curved Plate with Hole STIF 63 21

Chapter 3: Postprocessing 24

3.1 Pipe with cooling fins STIFF 55,70 24
3.2 Vector and Raster Contour Plots STIF 42 26
3.3 Animation of Cantilever Beam STIF 42 27
3.4 Path Operations STIF 42 28
3.5 Vector Data STIF 42 31
3.6 Line Element Processing STIF 3,12 32
3.7 Dump Truck Modal Analysis STIF 63 33
3.8 Interfacing Beams – Static Analysys STIF 3,12 36
3.9 Cantilever Beam with Distributed Load STIF 45 38
3.10 Symmetric Analysis – Cantilever Beam STIF 45 39
3.11 Error Norm Example – Elliptic Membrane STIF 42 40
3.12 Adaptive Meshing of Elliptic Membrane STIF 2 42

Chapter 4: Nonlinear Dynamics 46

4.1 Nonlinear Impulsively Loaded Clamped Plate STIF 43 46
4.2 Pipe Whip Analyssis Including Restraint STIF 20 47
4.3 Large Displacement Elastic Dynamic Response of Clamped Beam STIF 82 49
4.4 Impulsively Loaded Simply Supported Plastic Beam STIF 82 50
4.5 Slider – Crank Mechanisme Under Impact STIF 3,21,40 51

Chapter 5: The Four – Node Plastic Shell Element STIF 43 ..... **54**

5.1 Patch Test 54
5.2 Simply Supported .....Element Performance for Thick to Moderately Thi 55
5.3 Element Performance Fixed with Uniform Load 57
5.4 Element Performance : Simply Supported Distorted Mesh 59
5.5 Element Performance : Fixed , Distorted Mesh 61
5.6 Thick and Thin Cantilever Beam 63
5.7 Linear Analysis of a Cylindrical (Scordelis – LO) Shell 64
5.8 Linear Analysis of a Pinched Cylinder 65
5.9 Free Vibration of Simply Supported Rectangular Plate 66
5.10 Vibration of Prestressed Simply Supported Rectangular Plate 67
5.11 Vibration Behaveour of Twisted Cantilever Plates 68
5.12 Twisted Plate Vibration Under Centrifugal Loading 69
5.13 Buckling of a Hinged – Hinged Semi – Circular Arc 71
5.14 Buckling of a Closed Ring 72
5.15 Buckling of Hinged – Hinged Circular Cylindrical Shell 73
5.16 Large Deflection Analysis of a Clamped Square Plate 74
5.17 Simply Supported Square Plate Uniform Impulse Load 75
5.18 Nonlinear Impulsively Loaded Clamped Plate 76
5.19 Large Deflection Under Point Load of Hinged Spherical Shell 78
5.20 Nonlinear Analysis of a Barrel Vault Roof 79

Chapter 6: Solid Modeling **82**

6.1 Pinion Spur Gear, Contact Analysis (Model 1) STIF 82 82
6.2 Driven Spur Gear (Model 2) STIF 1,82 84
6.3 Spur Gear Combine Model 1 and Model 2 87
6.4 Generation Pass Spur Gear STIF 0,1 88
6.5 Use Pass Gear Model STIF 50,52 91
6.6 Stress Pass gear 92
6.7 Postprocessing Gear 93
6.8 Optimisation of a Fillet Radius STIF 83 94
6.9 Plastic Blower Wheel Housing STIF 93 99
6.10 Retaining Ring Model STIF 2 100
6.11 20 – Node Bricks Junct.with 10 – Node Tetrahedra(Top Model) STIF 92,95 102
6.12 8 – Node Brick Junction with 20 – Node Tetraheda(bot Model) STIF 45 103
6.13 Volume Generation with Continuous Mesh STIF 2 104
6.14 2 – Dimensional Contact Example STIF 0,1,12 105
6.15 3 – Dimensional Contact Example STIF 0,12,63,92 107
6.16 Connecting Dissimilar Meshes with Constraint Equation Interface 109
6.17 Primitive Geometry USER – FILE for Gear Tooth Profile 110

Chapter 7: Composite Structures **113**

7.1 Clamped Plate Analysis with Unsymmetrically Stacked Laminate STIF 99 113
7.2 Interlaminar Shear Stress in Thick Cantilever Beam STIF 46,99 115
7.3 Thick – Walled Press. Vessel Failure Optim.under int.pressure STIF 46 120
7.4 Buckling Analysis of Compos.Cylind. Panel with Circular Hole STIF 91 124
7.5 Documentation for Macro File LAYUP 127

7.6 Macro File LAYUP 128
7.7 Documentation for Macro File COLIST 133
7.8 Macro File COLIST 134

Chapter 8: Fracture Mechanics with ANSYS 135

8.1 Single – Edge Crack : Elastic Analysis STIF 2 135
8.2 Single – Edge Crack : Nodal Forces and Displacements STIF 2 136
8.3 Single – Edge Crack : Energy Difference Virt. Crack Ext. KI Comp STIF 2 137
8.4 Inclined Edge Crack : Elastic Analysis STIF 2 139
8.5 Compact Tension Specimen (Fine Model) J – Integral Path Evaluations 140
8.6 Compact Tension Specimen (Coarse Model) STIF 2 142
8.7 Compact Tension Specimen (Coarse Model) Using POST26 STIF 2 147
8.8 3 – D Semicircular Surface Flaw : Elastic Analysis STIF 92 149
8.9 Macro J – Integral User File for Compact Tension Specimen Analysis 153
8.10 Macro J – Integral User File for Single – Edge Crack Elasic Analysis 154

Chapter 9: Acoustic Pressure Wave Propagation ALL STIF 30 156

9.1 Pressure Step Propagation under Water 1 – D Wave 156
9.2 Oblique Plane Wave 157
9.3 Cylindrical Wave Propagation 159
9.4 Room Acoustics – Harmonic Analysis 160
9.5 Room Acoustics – Modal Analysis 163
9.6 Impedance Tube Problem – Harmonic and Modal Analysis 164

Appendix A: List of all Files on FLOPPY DISK from SWANSON Co. 167

A.1 Coupled Field Analysis 167
A.2 Revision 4.4 Changes Tutorial 167
A.3 Postprocessing 167
A.4 Nonlinear Dynamics 168
A.5 The Four – Node Plastic Shell Element STIF 43 168
A.6 Solid Modeling 168
A.7 Composite Structures 169
A.8 Fracture Mechanics using the ANSYS Program 169
A.9 Acoustic Pressure Wave Propagation 170

1. Coupled Field Analysis : Tutorial DN – T021

1.1 Tokamak Fusion Reactor Coil Analysis STIF 5

```
/PREP7
/TITLE, TOKAMAK FUSION REACTOR COIL ANALYSIS
KAN,0                          * KAN 0 STATIC ANALYSIS TYPE
ET,1,5,0,,,1                   * STIF5, DOF=UX,UY,UZ,VOLT,MAG,TEMP
KEYOPT,1,9,2                   * CALULATE SOURCE TERM 1ST TWO ITERATIONS
ET,2,5,9                       * STIF5, DOF=VOLT
KEYOPT,1,9,2                   * CALCULATE SOURCE TERM 1ST TWO ITERATIONS
KNL,1                          * NL TABLE FOR PROPERTY INPUT
NL,1,56,16                     * ROTATE CURRENT SOURCE TERMS ABOUT Z AXIS
NL,1,57,1                      * MKS UNITS
NL,2,55,1                      * RELATIVE PERMEABILITY FOR COIL
NL,2,103,6E7                   * ELECTRICAL CONDUCTIVITY OF COIL
NL,3,55,1                      * RELATIVE PERMEABILITY FOR COIL
NL,3,103,6E7                   * ELECTRICAL CONDUCTIVITY OF COIL
MP,EX,2,.68E11                 * RADIAL MODULUS OF ELASTICITY
MP,EY,2,.122E12                * LATERAL MODULUS OF ELASTICITY
MP,EZ,2,.122E12                * HOOP MODULUS OF ELASTICITY
MP,NUXY,2,.3                   * POISSON RATIO
/COM,
TDEG=11.25
TRAD=(11.25*3.141593)/180.
K,1,.526                       * DEFINE KEYPOINTS
K,51,.526,(.526*TAN(TRAD))
K,26,.88
K,76,.88,(.88*TAN(TRAD))
KGEN,2,1,76,1,,,.9,1
K,11,1.5,0,1.893 $RP2,50,,.187
K,36,1.5,0,1.480 $RP2,50,,.187
K,50,2.768,0 $RP2,50,,.187
K,25,3.181,0 $RP2,50,,.187
/COM
*CREATE,MAC                    * CREATE MACRO FOR LINE SEG. DEFINITION
SPLINE,ARG1,ARG2,,,,,,,-1,1
SPLINE,ARG2,ARG3,,,,,-1,,,,,-1
*END
*USE,MAC,2,11,25
RP4,,25,25,25
/COM
*CREATE,MC2                    * CREATE MACRO FOR LINE SEG. DIVISIONS
LDIV,ARG1,,,9
LDIV,ARG2,,,14
*END
*USE,MC2,1,2
RP4,,2,2
/COM
L,1,2,8 $L,1,26,4 $L,1,51,2
CSYS,0
V,1,26,76,51,2,27,77,52        * CREATE VOLUMES
```

```
V,2,27,77,52,3,28,78,53
RP23,1,1,1,1,1,1,1,1
VSYMM,2,ALL
VSYMM,3,ALL
*CREATE,MC3                    * CREATE MACRO FOR ATTRIBUTE ASSIGNMENT
CSKP,ARG1,0,ARG2,ARG3,ARG4     * INCLUDING TYPE, MAT, AND ESYS
VLSEL,,ARG5,(ARG5+24),24
VATT,2,,1,ARG1
VLALL
*END
*USE,MC3,11,26,1,76,1
RP24,,1,1,1,1,1
VLSEL,,49,96
VATT,3,,2
VLALL
ELSI,1,,2
VMESH,ALL                      * MESH VOLUMES
CSYS,0
NSEL,Z,0 $NINV                 * MERGE ALL NODES EXCEPT AT Z=0
MERGE
NSEL,Z,0 $NRSEL,X,0,1.         * MERGE TOP HALF OF NODES AT Z=0, INNER COIL
NRSEL,Y,0 $NRSEL,,1,485
MERGE
NSEL,Z,0 $NRSEL,X,0,1.         * MERGE BOTTOM HALF OF NODES AT Z=0, INNER COIL
NRSEL,Y,0 $NRSEL,,961,1445
MERGE
NSEL,Z,0 $NRSEL,X,0,1.         * SELECT TOP HALF NODES AT Z=0, INNER COIL
NRSEL,,1,495
CPSIZ,40
CP,1,VOLT,ALL                  * COUPLE VOLT DOF
HFLOW,1,AMPS,2.06E6            * APPLY CURRENT TO A NODE
NSEL,Z,0                       * SELECT BOTTOM HALF NODES AT Z=0, INNER COIL
NRSEL,X,0,1.
NRSEL,,961,1455
NT,ALL,VOLT,1.                 * FIX VOLT AT MID-PLANE
NSEL,Z,0 $NRSEL,X,2.5,3.5      * SELECT NODES AT Z=0, OUTER COIL
MERGE $NALL                    * MERGE TO COMPLETE PATH OF CURRENT FLOW
ESEL,TYPE,1 $NELEM
NT,ALL,MAG,0                   * FIX ALL MAG DOF TO ZERO (SOURCE FIELD ONY)
SYMBC,0,0,.526                 * APPLY STRUCTURAL B.C. AT VAULT (RADIAL)
CSYS,1
NSEL,Y,11.25
NASEL,Y,-11.25
NRSEL,X,.5363
D,ALL,UY,0                     * APPLY LATERAL CONSTRAINT AT VAULT
NALL
SYMBC,,3                       * APPLY VERTICAL B.C. AT MID-PLANE
NALL $EALL
CSYS,0
NSEL,Z,0 $NRSEL,X,0,1.
NRSEL,,1,495
WSTART,ALL                     * REORDER WAVEFRONT
NALL
```

```
WAVES
KBC,1                          * STEP B.C.'S
ITER,3,0,3                     * THREE ITERATIONS TO ALLOW FOR COUPLING
AFWRI
FINI
/INPUT,27
FINI
/OUT
/POST1
STRES,B,5,139                  * STORE FLUX DENSITY (MAGNITUDE)
STRES,JS                       * STORE CURRENT DENSITY VECTOR
STRES,BMAG                     * STORE FLUX DENSITY VECTOR
SET,1,3
/SHOW,FILE33
GLOBAL,0                       * SWITCH TO ELEMENT COORDINATE DISPLAY
/VIEW,,4,5,6
/TYPE,,3
/VUP,1,Z
/PNUM,TYPE,1
EPLOT                          * ELEMENT DISPLAY
ESEL,TYPE,1
NELEM
/SHOW,FILE33,,1
/TYPE,1
/NUM,1
/EDGE,1,1
PLVECT,JS                      * CURRENT DENSITY VECTOR DISPLAY
PLVECT,BMAG                    * FLUX DENSITY VECTOR DISPLAY
/SHOW,FILE33
/VIEW,1,,1
/VCONE,1,45
/TYPE,,3
/EDGE,1,0
/DIST,1,2.0
PLNS,SX                        * RADIAL STRESS DISPLAY
PLNS,SY                        * LATERAL STRESS DISPLAY
PLNS,SZ                        * HOOP STRESS DISPLAY
PLNS,SXZ                       * SHEAR STRESS DISPLAY
PLNS,B,1                       * FLUX DENSITY DISPLAY
/EDGE,1,1
PLDI,1                         * DISPLACED OVER UNDISPLACED DISPLAY
FINI
```

1.2 Hot Die Forging Thermal – Stress Analysis (STIF 13, STIF 34)

```
/PREP7
/TITLE, HOT DIE FORGING THERMAL-STRESS ANALYSIS
KAN,-1                                  * USE KAN,-1 ANALYSIS TYPE
ET,1,13,4,,1                            * STIF13, UX UY TEMP (MAG) DOF'S
ET,2,34                                 * STIF34 CONVECTION LINK
RSIZ,10
R1=(1/9)                                * DEFINE CONVECTION LINK REAL CONSTANTS
```

```
R,1,R1
RP6,1,R1
R,7,1.64 $R,8,2.03 $R,9,2.42
R,10,2.81 $R,11,3.94 $R,12,4.5
R,13,5.06 $R,14,5.63
MP,HF,1,69.4                          * EQUVALENT SURFACE CONTACT COEFFICIENT
ET,3,71                               * STIF71 LUMPED THERMAL MASS
R,15,6.11                             * HEAT CAPACITANCE
ET,4,31                               * STIF31 RADIATION LINK
R,16,3.75,1,-1,.119E-10               * REAL CONSTANTS FOR RADIATION LINK
R,17,2.73,1,.8,.119E-10
TOFFST,460                            * OFFSET TEMP FOR RADIATION
KNL,1
NL,2,49,0,392,1652                    * TEMPERATURES FOR CONDUCTIVITY
NL,2,97,7.0,8.08,15.08                * DIE CONDUCTIVITY VALUES
MP,DENS,2,.316                        * DIE DENSITY
MP,C,2,.095                           * DIE SPECIFIC HEAT
MPTEMP,1,70,1000,1400,1700            * TEMPERATURES FOR EMIS,EX,ALPX
MPDATA,EMIS,1,1,.54,.72,.79,.86       * DIE EMISSIVITY VALUES
MPDATA,EX,2,1,30.8E6,26.4E6,24.1E6,21.9E6       * DIE EX VALUES
MPDATA,ALPX,2,1,6.15E-6,7.42E-6,7.93E-6,8.31E-6  * DIE ALPX VALUES
/COM
TUNIF,70                              * SET INITIAL UNIFORM TEMPERATURE
K,1 $,2,2 $,3,4.5 $,4,7.5             * DEFINE KEYPOINTS
,5,,2 $,6,2,2 $,7,4.5,2 $,8,4.5,2,1
,9,7.5,2 $,10,,4 $,11,2.5,4 $,12,3,5,1
,13,,5 $,14,3,5 $,15,,5.5
,16,3,5.5 $,17,5.5,6 $,18,7.5,6
LOCAL,11,1,4.5,2
K,19,.5,225
CSYS,0
CIRCLE,7,.5,8,19,360,4                * DEFINE LINE SEGMENTS
CIRCLE,14,.5,12,16,-180,2
L2ANG,5,4,0,180
K,,4,4
L,11,29
LINT,9,10
LDIV,1
LANG,2,17,0
LDELE,5,7,2
LARC,16,27,14,.5                      * SET PARAMETERS FOR MESHING
N1=4
N2=6
N3=8
LMODIF,1,,,N1 $RP2,1                  * MODIFY LINE SEGMENTS
LMODIF,8,,,N1 $RP2,5
LMODIF,15,,,(N3+3)
LMODIF,11,,,N2,.35
LMODIF,9,,,N1 $LMODIF,5,,,(N1+1)
L,1,5,N1 $,2,6,N1 $,2,3,N1 $,3,4,N1
,4,9,N1 $,9,18,(N3+3) $,10,13,N1
,11,14,N1 $,13,15,(N1-1) $,14,16,(N1-1)
A,1,2,6,5 $,2,3,31,20 $,2,20,28,6     * CREATE AREAS
```

```
,3,4,21,31 $,4,9,32,21 $,32,9,18,17
,5,6,11,10 $,6,28,30,11 $,10,11,14,13
,11,30,27,14 $,13,14,16,15
ELSI,,N2,2
MAT,2                                   * MESH DIE
AMESH,ALL
A,14,27,16,16 $ELSI,,N2,1 $AMESH,12
N,400                                   * NODE FOR PLATEN LINK
N,401                                   * NODE FOR FURNACE LINK
TYPE,2 $MAT,1
REAL,1 $E,3,400                         * DEFINE CONVECTION LINKS
REAL,2 $E,4,400
REAL,3 $E,5,400
REAL,4 $E,6,400
REAL,5 $E,7,400
REAL,6 $E,2,400
REAL,7 $E,37,400
REAL,8 $E,38,400
REAL,9 $E,39,400
REAL,10 $E,36,400
REAL,11 $E,95,400
REAL,12 $E,96,400
REAL,13 $E,97,400
REAL,14 $E,94,400
TYPE,3 $REAL,15                         * DEFINE PLATEN
E,400
TYPE,4 $REAL,16                         * DEFINE RADIATION LINKS
E,94,401
E,122,401 $RP4,1
REAL,17
E,150,401 $RP11,1
NT,401,TEMP,1000                        * SET FURNACE WALL TO 1000 DEG.
LSSEL,,32,38,6 $LSASEL,,9,11,2 $LSASEL,,5
LSASEL,,1,8,7 $LSASEL,,2,13,11 $LSASEL,,15
LCVSF,ALL,1.5E-8,1E10 $LSALL            * APPLY HEAT FLUX (AS CONVECTION)
SYMBC                                   * APPLY STRUCTURAL CONSTRAINTS
SYMBC,,2
WSORT,X                                 * REORDER WAVEFRONT
KBC,1                                   * STEP BOUNDARY CONDITION
NUSEL,,400,401
NT,ALL,TEMP,70                          * FIX DIE TO 70 DEG.
NALL
TIME,0.0                                * DO SOLUTION AT TIME=0.0
ITER,-4,0,4                             * ITERATE FOR MAT. PROP. EVALUATION
LWRIT                                   * WRITE 1ST LOAD STEP
NUSEL,,400,401
NTDELE,ALL,TEMP                         * DELETE TEMP CONSTRAINT ON DIE
NALL
TIME,2.0                                * SET FOR TWO HOURS
ITER,-500,0,1                           * USE TIME STEP OPTIMIZATION
LWRIT                                   * WRITE 2ND LOAD STEP
AFWRI
FINI
```

```
/EXEC
/INPUT,27
FINI
/POST1
/SHOW,FILE33,,1
SETG                                    * SELECT ONLY DIE NODES AND ELEMENTS
NSEL,,400,401 $ENODE
NINV $EINV
SET,,,,,.26                             * SELECT SOLUTION AT TIME=.26 HOURS
PLNS,TEMP                               * DISPLAY TEMPERATURE PROFILE
PLNS,SIG1                               * DISPLAY SIG1 CONTOURS
SET,,,,,1.612                           * SELECT SOLUTION AT TIME=1.612 HOURS
PLNS,TEMP                               * DISPLAY TEMPERATURE PROFILE
PLNS,SIG1                               * DISPLAY SIG1 CONTOURS
FINI
/POST26
ESTR,2,118,89,SIG1   * NODE 133         * STORE ELEM 118, NODE 133 SIG1
ESTR,3,119,94,SIG1  * NODE 133          * STORE ELEM 119, NODE 133 SIG1
DISP,4,360,TEMP,T1                      * STORE TEMPS ALONG DIE SURFACE
DISP,5,380,TEMP,T2
DISP,6,40,TEMP,T3
DISP,7,161,TEMP,T4
/GRAPH,LABY,SIG1
PLVAR,2,3                               * DISPLAY TIME-VARYING SIG1
PRVAR,2,3                               * PRINT TIME-VARYING SIG1
/GRAPH,LABY,TEMP
PLVAR,4,5,6,7                           * DISPLAY TIME-VARYING TEMP DIST.
PRVAR,4,5,6,7                           * PRINT TIME-VARYING TEMP DIST.
FINI
```

1.3 Slot Embedded Conductor (STIF 13) Eddy Currents, Skin Effect

```
/PREP7
/TITLE, SLOT EMBEDDED CONDUCTOR
/COM,   FROM KONRAD, A. "INTEGRODIFFERENTIAL FINITE ELEMENT
/COM,   FORMULATION OF TWO-DIMENSIONAL STEADY-STATE SKIN
/COM,   EFFECT PROBLEMS", IEEE TRANS. MAGNETICS, VOL. MAG-18,
/COM,   NO.1, JAN 1982 PG. 284-292
/COM
/NOPR
KAN,3                                   * KAN 3 FULL HARMONIC SOLVER
ET,1,13                                 * STIF13, MAG DOF, (FOR AIR)
ET,2,13,6                               * STIF13, MAG VOLT DOF, (FOR CONDUCTOR)
KNL,1
NL,1,57,1                               * DEFINE MKS UNITS
NL,2,55,1                               * RELATIVE PERMEABILITY (CONDUCTOR)
COND=5.8005E7
NL,2,211,COND                           * DEFINE CONDUCTOR CONDUCTIVITY
K,1 $K,2,.00495                         * DEFINE KEYPOINTS
K,3,.0139 $K,4,.01885
KGEN,2,1,4,1,,.00845
KGEN,2,6,7,1,,.00855
```

```
KGEN,2,9,10,1,,.00645
A,1,2,6,5 $RP3,1,1,1,1          * DEFINE AREAS
A,6,7,10,9 $AATT,2,,2           * SET ATTRIBUTES FOR CONDUCTOR
A,9,10,12,11
ARSEL,,5 $AATT,1,,1 $ARALL      * SET ATTRIBUTES FOR AIR
ELSI,.00198
AMESH,ALL                       * MESH AREAS
CPSIZ,310                       * UP THE STORAGE FOR COUPLED DOF'S
ESEL,MAT,2 $NELEM               * SELECT ALL NODE IN CONDUCTOR
CP,1,VOLT,ALL                   * COUPLE ALL NODES IN VOLT
I=1.0                           * DEFINE TOTAL CURRENT
A=.235805E-3                    * DEFINE AREA OF CONDUCTOR
*GET,N1,NMIN                    * SELECT A NODE IN THE CONDUCTOR
HFLOW,N1,AMPS,I                 * APPLY 1 AMP TOTAL CURRENT
NALL $EALL
NSEL,Y,.02345                   * SELECT NODES AT TOP PLANE
NT,ALL,MAG,0 $NALL              * SET DIRICHLET B.C.
WSORT,X                         * REORDER WAVEFRONT
HARFRQ,45                       * SET OPERATING FREQUENCY
ITER,1,0,1                      * SET FOR 1 ITERATION
AFWRI
FINI
/EXEC
/INPUT,27
FINI
/POST1
STRES,JT,13,23                  * STORE TOTAL CURRENT DENSITY
STRES,JC,13,18                  * STORE SOURCE CURRENT DENSITY
STRES,JE,13,19                  * STORE EDDY CURRENT DENSITY
SET,1,1                         * STORE REAL COMPONENT SOLUTION
/PNUM,MAT,1
/SHOW,FILE33,,1
EPLOT                           * PLOT ELEMENTS
/EDGE,1,1
/NUM,1
PLNS,MAG                        * PLOT FLUX LINES
/NUM,0
ESEL,MAT,2
PLNS,JT,1                       * PLOT TOTAL CURRENT DENSITY
PLNS,JE,1                       * PLOT EDDY CURRENT DENSITY
*GET,JC,JC,1                    * GET REAL COMPONENT OF JS
ACRE=JC/(COND*I)                * CALCULATE AC RESISTANCE/LENGTH
SET,1,1,,1                      * BRING IN IMAGINARY DATA
*GET,JC,JC,1                    * GET IMAGINARY COMPONENT OF JS
ACRA=JC/(COND*I)                * CALCULATE AC REACTANCE/LENGTH
DCRE=(1/(COND*A))               * CALCULATE DC RESISTANCE/LENGTH
RAT=(ACRE/DCRE)                 * AC/DC LOSS RATIO
FINI
```

1.4 Double – Mass Loaded Transducer STIF 5

```
/PREP7
/TITLE, DOUBLE-MASS LOADED TRANSDUCER
KAN,2                              * KAN 2 MODE-FREQUENCY ANALYSIS
KAY,2,5                            * EXPAND FIRST 5 MODES
KAY,3,-5                           * NORMALIZE REDUCED MODE SHAPES
ET,1,5                             * STIF5, UX UY UZ VOLT (TEMP MAG) DOF'S
ET,2,5,2                           * STIF5, UX UY UZ DOF'S
MP,EX,2,2.76E11                    * MODULUS OF MASS
MP,DENS,2,17000                    * DENSITY OF MASS
MP,NUXY,2,.3                       * POISSON RATIO OF MASS
NL,3,103,7.1243E-9                 * E11 PERMITTIVITY OF RINGS
NL,3,157,7.1243E-9                 * E22 PERMITTIVITY OF RINGS
NL,3,211,5.8410E-9                 * E33 PERMITTIVITY OF RINGS
NL,3,138,10.5                      * E61 PIEZOELECTRIC CONSTANT, LOWER RING
NL,3,143,10.5                      * E52 PIEZOELECTRIC CONSTANT, LOWER RING
NL,3,145,-4.1                      * E31 PIEZOELECTRIC CONSTANT, LOWER RING
NL,3,146,-4.1                      * E32 PIEZOELECTRIC CONSTANT, LOWER RING
NL,3,147,14.1                      * E33 PIEZOELECTRIC CONSTANT, LOWER RING
NL,3,271,13.2E10,7.1E10,7.3E10     * INPUT [C] MATRIX OF RINGS
,3,277,13.2E10,7.3E10
,3,282,11.5E10
,3,286,3.0E10
,3,289,2.6E10
,3,291,2.6E10
MP,DENS,3,7500                     * DENSITY
NLCOPY,3,4
NL,4,138,-10.5                     * INPUT NEGATIVE [E] MATRIX, FOR UPPER RING
NL,4,143,-10.5
NL,4,145,4.1
NL,4,146,4.1
NL,4,147,-14.1
MP,EX,5,10E9                       * MODULUS OF ADHESIVE
MP,DENS,5,1700                     * DENSITY OF ADHESIVE
MP,NUXY,5,.38                      * POISSON RATIO OF ADHESIVE
/COM,
IRR=0.25                           * RING INNER RADIUS
ORR=0.375                          * RING OUTER RADIUS
IRM=0.2                            * MASS INNER RADIUS
ORM=0.425                          * MASS OUTER RADIUS
HR=0.125                           * RING HEIGTH
HM=0.5                             * MASS HEIGTH
/COM
CSYS,1
K,1,IRR,-2.5 $K,2,ORR,-2.5         * DEFINE KEYPOINTS
KGEN,2,1,2,1,,,HR
KGEN,2,3,4,1,,,.01
K,7,IRM,-2.5,((HR*2)+.01)
K,8,IRR,-2.5,((HR*2)+.01)
KGEN,2,7,8,1,(ORR-IRM)
KGEN,2,7,10,1,,,.01
KGEN,2,11,14,1,,,HM
```

```
KGEN,2,1,18,1,,5
KGEN,2,1,2,1,,,-.005
KGEN,2,19,20,1,,,-.005
L,1,19,1 $L,7,23,1                    * DEFINE LINE SEGMENTS
L,1,3,3 $L,5,8,3 $L,11,15,6
V,1,2,4,3,19,20,22,21 $VATT,3,,1      * CREATE VOLUMES, ASSIGN ATTRIBUTES
V,3,4,6,5,21,22,24,23
V,5,6,9,8,23,24,27,26
VLSEL,,3 $VATT,4,,1
V,8,9,13,12,26,27,31,30
V,37,38,2,1,39,40,20,19
VLSEL,,2,4,2
VLASEL,,5 $VATT,5,,2
V,11,12,16,15,29,30,34,33
RP3,1,1,1,1,1,1,1,1
VLSEL,,6,8 $VATT,2,,2
VLALL
ELSIZ,.03125,,2
VMESH,ALL                             * MESH VOLUMES
SYMBC,1,2,-2.5                        * APPLY STRUCTURAL B.C.'S
SYMBC,1,2,2.5
SYMBC,1,3,-.005
NSEL,Z,HR                             * SELECT NODES AT RING ELECTRODE SURFACES
NASEL,Z,HR+.01
NASEL,Z,0 $NASEL,Z,((HR*2)+.01)
NT,ALL,VOLT,0                         * SET VOLT TO ZERO FOR RESONANCE B.C.
NALL
M,ALL,UZ $M,ALL,UX                    * SELECT ALL RADIAL AND VERTICAL DOF'S (M'S)
LWRIT                                 * WRITE 1ST LOAD STEP
NSEL,Z,HR                             * SELECT NODES AT RING INTERFACE
NASEL,Z,HR+.01
NTDELE,ALL,VOLT                       * RELEASE VOLT CONSTRAINT FOR ANTI-RESONANCE B.C.
NALL
LWRIT                                 * WRITE 2ND LOAD STEP
MODMSH,DETACH                         * DETACH SOLID MODEL FROM MESH
NSCALE,,ALL,,,.0254,1,.0254           * SCALE MODEL FROM INCHES TO METERS
AFWRI
FINI
/INPUT,27
FINI
/POST1
/VIEW,1,1,1,1
/ANG,,-120
/TYPE,,2
/SHOW,FILE33,,1
LS=1
*CREATE,MAC                       * CREATE MACRO TO LOOP THROUGH MODE SHAPES
SET,LS,ARG1
PLDISP,1                            * PLOT MODE SHAPE OVER UNDISTORTED SHAPE
*END
*USE,MAC,1
RP4,,1
LS=2
```

```
*USE,MAC,1
RP4,,1
FINI
```

2. Revision 4.4 Changes Tutorial

2.1 Time Step Optimisation

```
/PREP7
/TITLE, TIME STEP OPTIMIZATION ANALYSIS WITH ST42
KAN,4                 * Nonlinear transient dynamic analysis
KAY,5,2               * KAN-4 with initial vel. & accel. zero
ALPHAD,0.0
BETAD,1.624975E-06    * Stiffness proportional damping
KNL,1                 * Nonlinear properties present
ET,1,42
ET,2,42
EX,1,30E6
DENS,1,0.00073        * Material density
NL,1,13,10            * Bilinear kinematic hardening
NL,1,19,1,100         * Temperature
NL,1,25,50000,50000   * Yield stresses
NL,1,31,6E6,6E6       * Second slope of stress-strain curve
C***                    Create mesh with STIF42
K,1,2.5
K,2,1.767766953,1.767766953
K,3,0,2.5
K,4,0,5.0
K,5,5.0,5.0
K,6,5
KLIST,ALL
CSYS,1
L,1,2,3
L,2,3,3
CSYS,0
L,3,4,5
L,4,5,3
L,5,6,3
L,6,1,5
L,2,5,5
LLIST,ALL
A,2,3,4,5
A,2,5,6,1
ALIST,ALL
AMESH,ALL
TYPE,2
EMODIF,26             * Isolate element 26
NLIST,ALL
ELIST,ALL
C***                    Apply boundary conditions
NSEL,X,-0.01,0.01
D,ALL,UX,0
NALL
NSEL,Y,-0.01,0.01
D,ALL,UY,0
NALL
```

```
C***                    Apply load
NSEL,Y,4.99,5.01
NLIST,ALL
PSF,ALL,,,-80000.0
NALL
C***                    First load step
KBC,0
TIME,0.00001,0.000003 * DTMAX = 0.000003
ITER,-10,0,1
POSTR,-1,1
LWRITE
C***                    Second load step
NSEL,Y,4.99,5.01
PDELE,ALL
NALL
KBC,1
TIME,0.001,0.000025   * DTMAX = 0.000025
ITER,-990,0,1
POSTR,-1,1
LWRITE
AFWRITE
FINISH
/EXEC
/INPUT,27
FINISH
/OUT
C***                    Post-process with POST26
/POST26
/NOPR
C***                    Write post data for nodal displ.
DISP,2,28,UX,UX28
DISP,3,1,UX,UX01
DISP,4,10,UY,UY10
DISP,5,5,UY,UY05
C***                    Write post data for element stresses
ESTR,6,26,16,SYJ
ESTR,7,-1,19,FREQ     * Response frequency
ESTR,8,-1,26,ITS      * Time increments
ESTR,9,-1,9,NEQB      * No. of equil. iterations
PRVAR,2,3,4,7,8
PRVAR,5,2,6,7,8
PRVAR,9
/SHOW,FILE33
C***                    Plot displ. and stress items
PLVAR,2,3,4,5
PLVAR,6
PLVAR,7
PLVAR,8
PLVAR,9
FINISH
```

2.2 Spin Softening

```
/PREP7
/TITLE,TEST SPIN SOFTENING
C*** MODAL ANALYSIS WITH NO LOAD. THEORY F1=23.8 HERTZ
C***                              ANSYS  F1=25.0 HERTZ
KAN,2
ET,1,45
EX,,217E9
DENS,,7850
N,1,-.014,.150
N,2,.014,.150
N,3,-.014,.478
N,4,.014,.478
NGEN,2,4,1,4,1,,,.003
N,9,-.014,.300
N,10,.014,.300
NGEN,2,2,9,10,1,,,.003
E,1,2,10,9,5,6,12,11
E,9,10,4,3,11,12,8,7
TOTAL,24
ITER,1,1,1
D,1,ALL
D,2,ALL
D,5,ALL
D,6,ALL
AFWRITE
FINISH
/INPUT,27
FINISH
C***
/PREP7
RESUME
C*** STATIC PRESTRESS FOR EDGE ROTATION
KAN,0
OMEGA,314.159265
AFWRITE
FINISH
/INPUT,27
FINISH
/FREQ,24,,,,1     *MODAL ANALYSIS WITH STRESS STIFFENING
OMEGA,314.159265,,,0    *RUN WITHOUT SPIN SOFTENING
ITER,1,1,1
END
FINISH
/FREQ,24,,,,1      *MODAL ANALYSIS WITH STRESS STIFFENING
OMEGA,314.159265,,,1      *RUN WITH SPIN SOFTENING
C*** MODAL ANALYSIS FOR EDGE ROTATION. THEORY F1=52.8 HERTZ
C***                                   ANSYS  F1=49.4 HERTZ
ITER,1,1,1
END
FINISH
```

2.3 Example to Compute Heat Loss Through Sides

```
/PREP7
/TITLE  EXAMPLE TO COMPUTE HEAT LOSS THRU SIDES
KAN,-1  $ET,1,55   $KXX,,1   $KYY,,10
N,1    $N,5,4    $FILL    $NGEN,9,10,1,5,1,,1
E,1,2,12,11    $EGEN,4,1,-1    $EGEN,8,10,-4
ITER,1,1   $CVSF,,, ,1    $CVSF,,,4,1
KTEMP,-1   $Q,2,1000   * POUR HEAT IN HERE AT NODE 2
/PBC,ALL,1
AFWR    $FINI
/EXEC
/INPUT,27
FINI
/POST1
STRESS,TFY,55,46   * SAVE THE VERTICAL FLUX AT THE ELEMENT CENTROIDS
SET,1,1            * HOLD RESULTS FROM FIRST ITERATION OF FIRST LOAD STEP
LPATH,21,25        * DEFINE PATH FOR LEVEL 1 BY GIVING END NODES
PDEF,INTR,L1,TFY   * INTERPOLATE VERTICAL FLUX TO PATH, CALLED L1
PRANGE,6           * LOOK AT EVERY FIFTH POINT
PVIEW,PRINT,L1
PVIEW,PLOT,L1
PCAL,INTG,F1,L1,S  * INTEGRATE FLUX ALONG LENGTH (CALL RESULT F1)
*GET,F1,PLAS,F1    * TAKE LAST PATH QUANTITY F1(VECTOR) AND CALL IT F1(SCALAR)
PDEF,SAVE          * HOLD F1 FOR LATER PLOTTING
LPATH,61,65
PDEF,INTR,L2,TFY
PVIEW,PRINT,L2
PVIEW,PLOT,L2
PVIEW,PLOT,L1,L2
PCAL,INTG,F2,L2,S
*GET,F2,PLAS,F2
LKPA = F1 - F2
*STAT
C***  COMPARE TO SUM OF APPLICABLE CONVECTION SURFACES
RESET
STRESS,CV2,55,19
STRESS,CV4,55,35
SET,1,1
NSEL,Y,2.5,5.5
ENODE
SSUM
*GET,CV2S,SSUM,CV2
*GET,CV4S,SSUM,CV4
LKCV = CV2S + CV4S
*STAT
FINISH
```

2.4 Slot Embedded Conductor

```
/PREP7
/TITLE, SLOT EMBEDDED CONDUCTOR
/COM,   FROM KONRAD, A. "INTEGRODIFFERENTIAL FINITE ELEMENT
/COM,   FORMULATION OF TWO-DIMENSIONAL STEADY-STATE SKIN
/COM,   EFFECT PROBLEMS", IEEE TRANS. MAGNETICS, VOL. MAG-18,
/COM,   NO.1, JAN 1982 PG. 284-292
/COM
/NOPR
KAN,3                                   * KAN 3 FULL HARMONIC SOLVER
ET,1,13                                 * STIF13, MAG DOF, (FOR AIR)
ET,2,13,6                               * STIF13, MAG VOLT DOF, (FOR CONDUCTOR)
KNL,1
NL,1,57,1                               * DEFINE MKS UNITS
NL,2,55,1                               * RELATIVE PERMEABILITY (CONDUCTOR)
COND=5.8005E7
NL,2,211,COND                           * DEFINE CONDUCTOR CONDUCTIVITY
K,1 $K,2,.00495                         * DEFINE KEYPOINTS
K,3,.0139 $K,4,.01885
KGEN,2,1,4,1,,.00845
KGEN,2,6,7,1,,.00855
KGEN,2,9,10,1,,.00645
A,1,2,6,5 $RP3,1,1,1,1                  * DEFINE AREAS
A,6,7,10,9 $AATT,2,,2                   * SET ATTRIBUTES FOR CONDUCTOR
A,9,10,12,11
ARSEL,,5 $AATT,1,,1 $ARALL              * SET ATTRIBUTES FOR AIR
ELSI,.00198
AMESH,ALL                               * MESH AREAS
CPSIZ,310                               * UP THE STORAGE FOR COUPLED DOF'S
ESEL,MAT,2 $NELEM                       * SELECT ALL NODE IN CONDUCTOR
CP,1,VOLT,ALL                           * COUPLE ALL NODES IN VOLT
I=1.0                                   * DEFINE TOTAL CURRENT
A=.235805E-3                            * DEFINE AREA OF CONDUCTOR
*GET,N1,NMIN                            * SELECT A NODE IN THE CONDUCTOR
HFLOW,N1,AMPS,I                         * APPLY 1 AMP TOTAL CURRENT
NALL $EALL
NSEL,Y,.02345                           * SELECT NODES AT TOP PLANE
NT,ALL,MAG,0 $NALL                      * SET DIRICHLET B.C.
WSORT,X                                 * REORDER WAVEFRONT
HARFRQ,45                               * SET OPERATING FREQUENCY
ITER,1,0,1                              * SET FOR 1 ITERATION
AFWRI
FINI
/EXEC
/INPUT,27
FINI
/POST1
STRES,JT,13,23                          * STORE TOTAL CURRENT DENSITY
STRES,JC,13,18                          * STORE SOURCE CURRENT DENSITY
STRES,JE,13,19                          * STORE EDDY CURRENT DENSITY
SET,1,1                                 * STORE REAL COMPONENT SOLUTION
/PNUM,MAT,1
```

```
/SHOW,FILE33,,1
EPLOT                                   * PLOT ELEMENTS
/EDGE,1,1
/NUM,1
PLNS,MAG                                * PLOT FLUX LINES
/NUM,0
ESEL,MAT,2
PLNS,JT,1                               * PLOT TOTAL CURRENT DENSITY
PLNS,JE,1                               * PLOT EDDY CURRENT DENSITY
*GET,JC,JC,1                            * GET REAL COMPONENT OF JS
ACRE=JC/(COND*I)                        * CALCULATE AC RESISTANCE/LENGTH
SET,1,1,,1                              * BRING IN IMAGINARY DATA
*GET,JC,JC,1                            * GET IMAGINARY COMPONENT OF JS
ACRA=JC/(COND*I)                        * CALCULATE AC REACTANCE/LENGTH
DCRE=(1/(COND*A))                       * CALCULATE DC RESISTANCE/LENGTH
RAT=(ACRE/DCRE)                         * AC/DC LOSS RATIO
FINI
```

2.5 Axisymmetric Flow – Flow between Coaxial Cylinders

```
/PREP7
/TITLE,UPDEX  AXISYMMETRIC FLOW -FLOW BETN. COAXIAL CYLINDERS  (CRK)
KAN,-1
ET,1,15,,,1
R,1
N,1,.2
N,5,.2,1.
FILL,1,5
NGEN,5,5,1,5,1,.1
E,1,6,7,2
EGEN,4,1,-1
EGEN,4,5,-4
DENS,1,1
VISC,1,5
KXX,1,1
CV,1,6,1.01E5,,16,5      * INLET PRESSURE
CV,10,5,1.00E5,,25,5     * EXIT PRESSURE
NT,1,TEMP,,,25,1
D,1,UX,,,5,1,UY          * NO SLIP BOUNDARY CONDITION AT INNER WALL
D,21,UX,,,25,1,UY        * NO SLIP BOUNDARY CONDITION AT OUTER WALL
D,6,UX,,,16,5            * AXIAL INLET FLOW
D,10,UX,,,20,5           * AXIAL EXIT FLOW
ITER,-5,5,5
KBC,1
AFWRITE
FINISH
/INPUT,27
FINISH
```

2.6 Thermal Effects in HAGEN – POISEUILLE Flow

```
/PREP7
/SHOW,FILE33,DAT
/TITLE, EV15-12, PGA, THERMAL EFFECTS IN HAGEN-POISSEULLE FLOW
/COM, --- AXISYMMETRIC FLOW THROUGH A LONG PIPE IS MODELED
/COM, --- PIPE IS SUBJECTED TO UNIFORM HEAT FLUX ALONG WALL
/COM, --- VELOCITY PROFILE IS FULLY DEVELOPED THROUGHOUT LENGTH
/COM, --- ENTRANCE TEMPERATURE IS UNIFORM ACROSS INLET AREA
/COM, --- TEMP. AT OUTLET IS EXPECTED TO REACH FULL DEVELOPMENT
/COM, --- ENTRANCE LENGTH AND FULLY DEVLPD PROFILE IS STUDIED
/COM, --- REF: CONVECTION HEAT TRANSFER BY ADRIAN BEJAN, 1984
/NOPR
KAN,-1
ET,1,15,,,1,,,,1
R,1                    *** TRANSPORT PROPERTIES OF WATER ***
KXX,1,0.0058           * K=0.0058; DENS=1.0; SP.HT=4.217   *
DENS,1,1.0             * VISC=0.01787;                     *
C,1,4.217              * UNITS: HEAT-J; LENGTH-CM; WT-GM   *
VISC,1,0.01787         ****************************************
N,1
N,9,2.5                  *** GEOMETRY : L=16.0; DIA=5.0 ****
FILL
NGEN,41,10,1,9,1,,0.4    *** MESH : 8 X 40              ****
E,1,2,12,11
EGEN,8,1,-1
EGEN,40,10,-8
D,ALL,UX
NSEL,X,2.5
D,ALL,UY
NUSEL,Y
NUSEL,Y,16
HFLOW,ALL,HEAT,10       * UNIFORM WALL HEAT FLUX OF 10.0 (J/S-CM**2)
NSEL,X,2.5              * END NODES HAVE HALF THE HEAT FLOW (PER RADN)
NUSEL,Y,0.1,15.9
HFLOW,ALL,HEAT,5
NSEL,Y
CVSF,ALL,,,1.10979E-3   * PRESSURE DIFFERENTIAL FOR UNIFORM FLOW
NT,ALL,TEMP,50          * UNIFORM ENTRANCE TEMPERATURE T-IN=50
NSEL,Y,16
CVSF,ALL,,,0.0
NALL
KTEMP,-1
TUNIF,50
ITER,-10,0,10
KBC,1
AFWRITE
FINI
/INP,27
FINI
/POST1
SET,1,10
NSEL,Y
```

```
PRDISP
/TITLE  INLET VELOCITY PROFILE (UNIFORM ALONG LENGTH)
PLPATH,1,9,UY
/COM, **********
/COM, <<< FULLY DEVELOPED VEL PROFILE & UNIFORM TEMP AT Y=0 (INLET)>>>
/COM, **********
NSEL,Y,10
PRDISP
/TITLE  FULLY DEVELOPED TEMPERATURE PROFILE AT C-C
PLPATH,251,259,TEMP
/COM, **********
/COM, <<< FULLY DEVELOPED VEL & TEMP PROFILES AT Y=10 (NEAR OUTLET)>>>
/COM, **********
NSEL,X
PRDISP
/TITLE  TEMPERATURE VARIATION ALONG CENTERLINE (X=0)
PLPATH,1,281,TEMP
/COM, **********
/COM, <<< TEMPERATURE VARIATION ALONG CENTERLINE (X=0) >>>
/COM, **********
NSEL,X,2.5
PRDISP
/TITLE  TEMPERATURE VARIATION ALONG TUBE WALL (X=2.5)
PLPATH,9,289,TEMP
/COM, **********
/COM, <<< TEMPERATURE VARIATION ALONG TUBE WALL (X=2.5) >>>
/COM, **********
FINI
```

2.7 Test STIF 43 Elastically

```
/PREP7
THCK = .1
/TITLE   TEST STIF43 ELASTICALLY
ET,1,43,,, ,,, ,1
ET,2,63,,, ,,, ,1
ET,3,45,,, ,,, ,1
R,1,THCK
C****   COMPUTE ACTUAL VALUES USING PARAMETRIC MATH
B=3
LEN=20
EX=30E6
DZ=.01
I = (B*(THCK**3))/12
AS = (B*THCK)*(5/6)
C**** REACTION FORCE (P) = ((DZ*EX)/LEN)/(((LEN**2)/(3*I)) + (2/AS))
P = ((DZ*EX)/LEN)/(((LEN**2)/(3*I)) + (2/AS))
P2=P/2
P4=P/4
/NOPR
EX,,30E6
NUXY,,0
```

```
N,1
N,2,,3
NGEN,5,2,1,2,1,5
NGEN,4,10,1,10,1,,,THCK
E,1,3,4,2
EGEN,4,2,-1
TYPE,2
E,11,13,14,12
EGEN,4,2,-1
TYPE,3
E,21,23,24,22,31,33,34,32
EGEN,4,2,-1
ITER,1,1
NSEL,X
D,ALL,ALL
NALL
NSEL,X,20
D,ALL,ROTX
D,ALL,UZ,.01
NALL
AFWR
FINI
/INPUT,27
FINI
```

2.8 Optimisation of Fibre Angle of Composite Material

```
ANG=45                          * FIBER DIRECTION ANGLE
/PREP7
/TITLE   DEMONSTRATION OF OPTIMIZATION OF FIBER ANGLE OF COMPOSITE MATERIAL
/COM  REFERENCE:  COMPOSITES DESIGN, S. W. TSAI, 1987
/COM  EXPECTED OPTIMUM ANGLE = 55.5, FROM PAGE 23-6
/OUT,26                         *  DIVERT BULK OF OUTPUT TO ANOTHER FILE
THK=1                           * WALL THICKNESS
RI=100                          * INNER RADIUS
PHI=3.0                         * SUBTENDED ANGLE
PRES=0.5                        * INTERNAL PRESSURE
RO =(100+THK)                   * OUTER RADIUS
PI = 3.1415926535
LEN = (PI/180)*(PHI*RI)         * AXIAL ELEMENT LENGTH
F=(PI*(RI*RI))                  * AXIAL FORCE
AREA=(PI*((RO*RO)-(RI*RI)))     * AREA OF CROSS-SECTION
F=((F*PRES)/AREA)               * AXIAL PRESSURE
ET,1,46                         * SELECT STIF46
R,1,2   $RMORE                  * REAL CONSTANTS
RMORE,1,ANG,(THK/2),1,-ANG,(THK/2)
C***     INPUT MATERIAL PROPERTIES
C***     USE CFRP (CARBON FIBER REINFORCED PLASTIC)   T300/N5208
C***     TAKEN FROM PAGE 6-8 OF REFERENCE
EX,1,181E3   $EY,1,103E3     * YOUNG'S MODULI
PRXY,1,.28    $PRYZ,1,0      $PRXZ,1,0   * POISSON'S RATIOS
```

```
GXY,1,7.17E3                * SHEAR MODULUS
NL,1,85,,,1                 * SELECT TSAI-WU FAILURE CRITERION
NL,1,97,9999                *  NON-TEMPERATURE DEPENDENT FAILURE CRITERIA
NL,1,157,1500                 * FAILURE STRESS, X  DIRECTION TENSION
NL,1,163,-1500                 * FAILURE STRESS, X  DIRECTION COMPRESSION
NL,1,169,40                   * FAILURE STRESS, Y  DIRECTION TENSION
NL,1,175,-246                  * FAILURE STRESS, Y  DIRECTION COMPRESSION
NL,1,181,40                   * FAILURE STRESS, Z  DIRECTION TENSION
NL,1,187,-246                  * FAILURE STRESS, Z  DIRECTION COMPRESSION
NL,1,193,68                   * FAILURE STRESS, XY  DIRECTION SHEAR
NL,1,199,100000               * LARGE NUMBER FOR YZ DIRECTION SHEAR
NL,1,205,100000               * LARGE NUMBER FOR XZ DIRECTION SHEAR
NL,1,211,-1                   * XY TSAI-WU COUPLING COEFFICIENT (DEFAULT)
NL,1,217,-1                   * YZ TSAI-WU COUPLING COEFFICIENT (DEFAULT)
NL,1,223,-1                   * XZ TSAI-WU COUPLING COEFFICIENT (DEFAULT)
C***     END OF MATERIAL PROPERTIES
CSYS,1                       * SELECT CYLINDRICAL SYSTEM
N,1,RI $N,2,RI,PHI  $N,3,RI,PHI,LEN  $N,4,RI,,LEN    *LOCATE NODES ON INSIDE
NGEN,2,4,1,4,1,THK           * GENERATE NODES ON OUTSIDE
NROTAT,ALL        * ROTATE NODAL COORDINATE SYSTEMS TO CYLINDRICAL
E,2,3,4,1,6,7,8,5            * DEFINE ELEMENT
SYMBC,1,2    $SYMBC,1,2,PHI   $SYMBC,1,3    * DEFINE PLANES OF SYMMETRY
EP,1,1,PRES  $EP,1,3,-F      * APPLY PRESSURES
CP,1,UZ,3,4,7,8              * COUPLING USED TO MAKE ONE ELEMENT MODEL VALID
CP,2,UX,1,2,3,4     $CP,3,UX,5,6,7,8
ITER,1,1,1 $AFWR  $FINI      * FINISH PREP7
/INP,27
FINI
/POST1
STRESS,FC3,46,58  $SET
*GET,FVAL,FC3,1  * STORE THE VALUE OF THE FAILURE CRITERION
FINI
/OPT
OPVAR,ANG,DV,00,90,0.01  * ALLOW ANGLE TO VARY THRU ENTIRE QUADRANT
C***  WATCH FOR LOCAL MINIMUM AT ZERO DEGREES.  THIS IS NOT WANTED.
OPVAR,FVAL,OBJ,,,0.0001 * GIVE TIGHT TOLERANCE ON OBJECTIVE FUNCTION
OPEQN,,,,1    * GET FITTING DATA
OPCOPY
OPRUN,20,,,7  * USE 7 RANDOM SOLUTIONS TO TRY TO AVOID LOCAL MINIMUM
/OUT
OPLIST,ALL
/OUT,26
C***   REFINE SOLUTION
OPSEL,5       * SELECT FIVE BEST SOLUTIONS, AND DISCARD THE REST
OPRUN,10
/OUT
OPLIST,ALL
/OUT,26
C***   REFINE SOLUTION AGAIN
OPSEL,5       * SELECT FIVE BEST SOLUTIONS, AND DISCARD THE REST
OPRUN,10
/OUT
OPLIST,ALL
```

```
FINI
```

2.9 Test STIF 26 with three Nodes and Large Rotation

```
/PREP7
/TITLE     TEST STIF26 WITH THREE NODES AND LARGE ROTATIONS
N,1,3,5
N,2
N,3,20,-20
R,1,100,-20
ET,1,26
E,1,3,2    * ORDER DIFFERENT THAN TUTORIAL NOTES
KAY,6,1    * LARGE DISPLACEMENTS
ITER,-3,1
D,1,ALL
D,2,ALL
D,3,UX
D,3,UY,40
AFWR
FINI
/INPUT,27
FINI
```

2.10 Curved Plate with Hole

```
/COM,ANSYS REVISION  4.4    UP414           4.6408     2/22/1989
/SHOW,CSYSDEMO,PIC,1
/PREP7
/TITLE   CURVED PLATE WITH HOLE
/VIEW,1,1,2,3
/PNUM,LINE,1
/PNUM,KPOI,1
LOCAL,11,1,0,0,20,20,-90,0
DSYS,11
K,
K,, , ,120
K,,1,
KLIST
CIRCLE,1,12,2,3
LPLO
KPLO
LPLO
CSYS,
L,     1,     2
ADRA,1,2,3,4,,,5
APLO
APLO,1
APLO,2
CSYS,1
K,,100,
K,,100,,50
```

```
KGEN,2,12,13,1,,45
A,    12,    13,    15,    14
LAINT,ALL
NUMMRG,KPOI
ADEL,1,4,,1
ALIST
ADEL,5
KPLO
KDEL,     3
LDEL,5,,,1
KPLO
LPLO
/PNUM,KPOI,0
/PNUM,LINE,0
/TIT          EDGE PLOT OF MODEL
LPLO
/PNUM,KPOI,1
/PNUM,LINE,1
CSLIST
/VIEW,1,.94,.34
LPLO
CSYS,1
LDEL,17
KGEN,2,16,16,1,0,0,-8
L,1,12
L,14,1
L,16,1
K,2,100,,20
LDEL,14
L,12,2
L,13,2
L,21,2
AL,1,3,21,6,4
/PNUM,AREA,1
APLO
LPLO
LSUS,LINE,          1
LSUS,LINE,4
LSUS,LINE,         21
LPLO
AL,ALL
LSALL
KPALL
/TIT      EDGE PLOT OF MODEL VIEWED PARALLEL TO HOLE AXIS
APLO
ET,1,63
R,1,1
EX,1,2
EY,1,1
KDVS,ALL,10
KPSE,,16,21
KPLO
KDVS,ALL,3
```

```
LDVA
KPALL
LPLO
LDVS,3,,3
AMESH,1
DSYS
/SHRIN,.1
/PSYM,ESYS,1
/TIT    AREA 1 WITH RANDOM ORIENTATIONS
EPLO
ACLEAR,1
ESYS,11
AMESH,1
/TIT   AREA 1 ORIENTED WITH LOCAL CYLINDRICAL
EPLO
AMESH,2
/TIT    ENTIRE MODEL ORIENTED WITH LOCAL CYLINDRICAL
EPLO
ACLEAR,ALL
LOCAL,12,1
AATT,1,1,1,12
AMESH,ALL
/TIT   ENTIRE MODEL ORIENTED WITH GLOBAL CYLINDRICAL
EPLO
/VIEW,1,1,2,3
EPLO
AFWR,,1
FINI
```

3. Postprocessing

3.1 Pipe with cooling fins

```
/PREP7
/TITLE,PIPE WITH COOLING FINS - 2D AND 3D THERMAL ANALYSIS
KAN,-1                      * Thermal analysis
ET,1,70                     * 3D thermal solid elements
ET,2,55,,,1                 * 2D axisymmetric thermal solid elements
MP,KXX,1,25.961             * Thermal conductivity in W/M-DEG C
MP,EX,1,193044              * Young's modulus in N/SQ.M xE06 (MPa)
MP,ALPX,1,1.62E-5           * Expansion coefficient (1/DEG C)
LOCAL,11,1,,,,0,22.5,-90    * CSYS,11 is cylindrical about global Y
K,1,0.127                   * Inside radius = 0.127 M. = 5 IN.
K,2,0.1524                  *  Pipe radius = 0.1524 M. = 6 IN.
K,3,0.3048                  *   Fin radius = 0.3048 M. = 12 IN.
KGEN,2,1,3,1,,,0.00635      * Fin half thickness = 0.00635 M. = 0.25 IN.
KGEN,2,1,2,1,,,0.0254       * Model height = 0.0254 M. = 1 IN.
L,1,2,8,4
L,2,3,10,8
L,4,5,8,4
L,5,6,10,8
L,7,8,8,4
L,1,4,1
RP3,1,1
L,4,7,3
RP2,1,1
LGEN,2,1,10,1,,45
L,1,9,10
RP8,1,1
A,7,8,16,15                 * Area 1 = periodic symmetry plane at Z=0.0254 M.
A,1,2,5,4                   * Areas 2-4 = symmetry plane at THETA=0
A,2,3,6,5                   *
A,4,5,8,7                   *
A,9,10,13,12                * Areas 5-7 = symmetry plane at THETA=45
A,10,11,14,13               *
A,12,13,16,15               *
A,1,2,10,9                  * Areas 8-9 = periodic symmetry plane at Z=0
A,2,3,11,10                 *
A,1,4,12,9                  * Areas 10-11 = inside wall
A,4,7,15,12                 *
A,3,6,14,11                 * Areas 12-14 = outside of pipe and fin
A,5,6,14,13                 *
A,5,8,16,13                 *
V,1,2,5,4,9,10,13,12
V,2,3,6,5,10,11,14,13
V,4,5,8,7,12,13,16,15
ACVSF,10,817.7,232.2        * Inside convection boundary
ACVSF,11,817.7,232.2        *  817.7 W/SQ.M-DEG C, 232.2 DEG C
ACVSF,12,204.4,21.1         * Outside convection boundary
ACVSF,13,204.4,21.1         *  204.4 W/SQ.M-DEG C,  21.1 DEG C
ACVSF,14,204.4,21.1         *
```

```
VMESH,ALL
WSORT,X                    * Reorder for solution efficiency
TYPE,2                     * Begin construction of 2D model
CSYS,0                     * Create solid model in the global X-Y plane
K,17,0.127                 * Inside radius = 0.127 M. = 5 IN.
K,18,0.1524                *  Pipe radius = 0.1524 M. = 6 IN.
K,19,0.3048                *   Fin radius = 0.3048 M. = 12 IN.
KGEN,2,17,19,1,,0.00635    * Fin half thickness = 0.00635 M. = 0.25 IN.
KGEN,2,17,18,1,,0.0254     * Model height = 0.0254 M. = 1 IN.
L,17,18,8,4
L,18,19,10,8
L,20,21,8,4
L,21,22,10,8
L,23,24,8,4
L,17,20,1
RP3,1,1
L,20,23,3
RP2,1,1
A,17,18,21,20
A,18,19,22,21
A,20,21,24,23
LCVSF,34,817.7,232.2       * Inside convection boundary
LCVSF,37,817.7,232.2       *
LCVSF,38,204.4,21.1        * Outside convection boundary
LCVSF,32,204.4,21.1        *
LCVSF,36,204.4,21.1        *
AMESH,17,19
ARSEL,,17,19,1,1
WSORT,X                    * Reorder 2D elements
VLALL                      * Select both 2D and 3D models
ARALL                      *
LSALL                      *
KPALL                      *
NALL                       *
EALL                       *
AFWRIT
FINISH
/GET,12,THERMAL,12         * Save thermal analysis results on file: THERMAL.12
/EXEC
/INPUT,27
FINISH
/PREP7
RESUME                     * Resume thermal model
SBCDEL,ACVSF,10            * Delete 3D solid model thermal B.C.
SBCDEL,ACVSF,11            *
SBCDEL,ACVSF,12            *
SBCDEL,ACVSF,13            *
SBCDEL,ACVSF,14            *
SBCDEL,LCVSF,32            * Delete 2D solid model thermal B.C.
SBCDEL,LCVSF,34            *
SBCDEL,LCVSF,36            *
SBCDEL,LCVSF,37            *
SBCDEL,LCVSF,38            *
```

```
/TITLE,PIPE WITH COOLING FINS - 2D AND 3D THERMAL STRESS ANALYSIS
KAN,0                    * Change from thermal to static analysis
CSYS,11
ARSEL,,2,4,1,1
NROTAT,ALL
D,ALL,UY                 * Apply symmetry B.C. on 3D model at THETA=0
ARSEL,,5,7,1,1
NROTAT,ALL
D,ALL,UY                 * Apply symmetry B.C. on 3D model at THETA=45
ARSEL,,8,9,1,1
NROTAT,ALL
D,ALL,UZ                 * Restrain base of 3D model with symmetry B.C. at Z=0
ARSEL,,1,1,1,1
NROTAT,ALL
CPSIZE,100
CP,1,UZ,ALL              * Enforce periodic symmetry assumption with CDOF
CSYS,0
LSSEL,,29,30,1,1
D,ALL,UY                 * Restrain base of 2D model at Y=0
LSSEL,,33,33,1,1
CP,2,UY,ALL              * Enforce periodic symm. assumption on 2D model
VLALL                    * Select both 2D and 3D models
ARALL                    *
LSALL                    *
KPALL                    *
NALL                     *
EALL                     *
APSF,10,6.8944           * Pressure load on inside wall of 3D model
APSF,11,6.8944           *   6.8944 MPa = 6.8944 xE06 N/SQ.M = 1000 PSI
LPSF,34,6.8944           * Pressure load on inside wall of 2D model
LPSF,37,6.8944           *
LWRITE                   * Load step 1 = inside pressure only
KTEMP,1,1                * Include temperature load from thermal analysis
LWRITE                   * Load step 2 = add temperatures to loading
AFWRIT
FINISH
/GET,12,STRESS,12        * Save stress analysis results on file: STRESS.12
/EXEC
/INPUT,27
FINISH
```

3.2 Vector and Raster Contour Plots

```
/GET,12,STRESS,12,[DJ.TUT44.SECT1_EX]
/SHOW,FILE33,,1      * VECTOR DATA ON FILE33
/POST1
STRESS,CSY,42,2      * ELEMENT ITEM 2 FOR STIF42 IS CENTROID SY
SET,2,1
/FOCUS,1,0.15,0.009  * SET DISPLAY TO ZOOM IN ON THE CORNER
/DIST,1,0.01         *  BETWEEN THE PIPE AND FIN
ESEL,STIF,42         * SELECT ONLY STIF42 ELEMENTS
NELEM                *  AND ATTACHED NODES
```

```
/TITLE,AVERAGED NODAL STRESS EXAMPLE
PLNSTR,SY
/TITLE,UNAVERAGED NODAL STRESS EXAMPLE
PLNSTR,SY,1
/TITLE,AVERAGED ELEMENT STRESS EXAMPLE
PLNSTR,CSY,1
/TITLE,UNAVERAGED ELEMENT STRESS EXAMPLE
PLNSTR,CSY
/PNUM,SVAL,1          * ADD DATA VALUES TO CONTOUR DISPLAY
/TITLE,UNAVERAGED ELEMENT STRESS EXAMPLE WITH SVAL
PLNSTR,CSY
/TITLE,AVERAGED NODAL STRESS EXAMPLE WITH SVAL
PLNSTR,SY
/PNUM,SVAL,0          * REMOVE DATA VALUES FROM CONTOUR DISPLAY
/SHOW,FILE33          * RASTER DATA ON FILE33 (FOR OVERHEAD TRANSPARENCIES)
/TITLE,AVERAGED NODAL STRESS EXAMPLE
PLNSTR,SY
/TITLE,UNAVERAGED NODAL STRESS EXAMPLE
PLNSTR,SY,1
/TITLE,AVERAGED ELEMENT STRESS EXAMPLE
PLNSTR,CSY,1
/TITLE,UNAVERAGED ELEMENT STRESS EXAMPLE
PLNSTR,CSY
/PNUM,SVAL,1          * ADD DATA VALUES TO CONTOUR DISPLAY
/TITLE,UNAVERAGED ELEMENT STRESS EXAMPLE WITH SVAL
PLNSTR,CSY
/TITLE,AVERAGED NODAL STRESS EXAMPLE WITH SVAL
PLNSTR,SY
FINISH
```

3.3 Animation of Cantilever Beam

```
/SHOW,4207,,1          * REQUEST VECTOR GRAPHICS
/PREP7
/TITL,CANTILEVER BEAM WITH UNIFORM PRESSURE LOAD
ET,1,42                * PLANE STRESS STIF42
MP,EX,1,3E7               * YOUNGS MODULUS
N,1
N,4,,1
FILL
NGEN,10,5,1,4,1,.5     * BUILD MODEL ALONG X AXIS IN THE X-Y PLANE
E,1,2,7,6
EGEN,3,1,1
EGEN,9,5,-3
D,1,UX,,,4             * FIX LEFT END OF BEAM
D,1,UY
PSF,0,2,1,100          * APPLY UNIFORM PRESSURE LOAD ALONG TOP OF BEAM
AFWRIT
FINISH
/INPUT,27
FINISH
/POST1
```

```
SET,1,1
/PLOFF,1,1,1,1          * DEACTIVATE EVERYTHING IN THE DISPLAY EXCEPT THE MODEL
/EDGE,1,1               * REMOVE INTERIOR ELEMENT OUTLINES ON PLOTS
*CREATE,MAC             * MACRO FOR DEFORMED SHAPE DISPLAY WITH
/DSCALE,1,ARG1          * CHANGING /DSCALE BY MACRO ARGUMENT 1
PLDISP                  * PLOT DEFORMED SHAPE
*END
/SEG,MULTI              * ACTIVATE MULTIPLE LOCAL SEGMENT STORAGE
*USE,MAC,500            * EXECUTE THE MACRO WITH /DSCALE VALUES OF
RP6,,-200               *      500, 300, 100, -100, -300, -500
/SEG,OFF                * DEACTIVATE MULTIPLE LOCAL SEGMENT STORAGE
ANIM,5                  * RUN ANIMATION THROUGH 5 CONTINUOUS CYCLES
/SEG,DELETE             * DELETE ALL LOCAL SEGMENTS
FINISH
/POST1
LCLIM,2                 * ALLOW 2 LOAD CASES FOR STRESS SCALING
SET,1,1                 * STORE LOAD CASE 1 DATA FROM LOAD STEP 1, ITER 1
/PLOFF,1,1,1,1          * DEACTIVATE EVERYTHING IN THE DISPLAY EXCEPT THE MODEL
/EDGE,1,1               * REMOVE INTERIOR ELEMENT OUTLINES ON PLOTS
/DSCALE,1,500           * SET /DSCALE FACTOR AND CONTOUR INTERVALS FOR ALL PLOTS
/CVAL,1,-4500,-3000,-1000,-500,500,1000,3000,4500
*CREATE,MAC2            * MACRO FOR CONTOUR DISPLAY LOOPING
LCFACT,ARG1             *  FACT1 = MACRO ARGUMENT 1
LCADD,2,1               *  CREATE LOAD CASE 2 FROM LOAD CASE 1 * FACT1
LCASE,2                 *  ACTIVE LOAD CASE IS NUMBER 2
PLNSTR,SX               * PRODUCE SX CONTOUR PLOT
*END
/SEG,MULTI              * ACTIVATE MULTIPLE LOCAL SEGMENT STORAGE
*USE,MAC2,1             * EXECUTE THE MACRO WITH VALUES FOR FACT1 OF
RP6,,-0.4               *     1.0, 0.6, 0.2, -0.2, -0.6, -1.0
/SEG,OFF                * DEACTIVATE MULTIPLE LOCAL SEGMENT STORAGE
ANIM,5                  * RUN ANIMATION THROUGH 5 CONTINUOUS CYCLES
/SEG,DELETE             * DELETE ALL LOCAL SEGMENTS
FINISH
```

3.4 Path Operations

```
/SHOW,PLSECT,F33,1      * PUT DISPLAYS ON A FILE33
/GET,12,STRESS,12,[DJ.TUT44.SECT1_EX]  * ATTACH FILE12 CONTAINING STRESS RESULTS
/POST1
SET,2,1                 * READ DATA FROM LOAD STEP 2 ITERATION 1
ESEL,STIF,42            * FIRST WORK WITH STIF42 AXISYMMETRIC ELEMENTS
NELEM                   *     AND THE NODES ASSOCIATED WITH THE STIF42
NRSEL,Y,0               * SELECT AND *GET NODES ON INSIDE WALL (N42A)
NRSEL,X,0.127           *  AND AT TIP OF FIN (N42B) ON THE MID-PLANE
*GET,N42A,NDMN          *  OF THE FIN FOR USE IN PATH OPERATIONS.
NELEM                   * SELECT AGAIN ALL NODES ON STIF42 ELEMENTS
NRSEL,Y,0
NRSEL,X,0.3048
*GET,N42B,NDMN
NELEM                   * SELECT AGAIN ALL NODES ON STIF42 ELEMENTS
NRSEL,Y,0               * ALSO SELECT AND *GET THE NODE ON THE MID-PLANE
```

```
NRSEL,X,0.1524          *  OF THE FIN AT THE RADIUS OF THE PIPE O.D. TO
*GET,N42C,NDMN          *  USE AS AN INTERMEDIATE PATH NODE, LATER.
CSYS,11                 * ACTIVATE CSYS,11 TO WORK WITH 3D MODEL
ESEL,STIF,45            * NEXT WORK WITH STIF45 BRICK ELEMENTS
NELEM                   *  AND THE NODES ASSOCIATED WITH THE STIF45
NRSEL,Z,0               * SELECT AND *GET NODES ON INSIDE WALL (N45A)
NRSEL,Y,22.5            *  AND AT TIP OF FIN (N45B) AT THE MID-PLANE
NRSEL,X,0.127           *  OF THE FIN (Z=0.0) AND AT THE MIDDLE SECTION
*GET,N45A,NDMN          *  OF THE 45 DEGREE SECTOR MODEL (THETA=22.5).
NELEM                   * THESE NODE NUMBERS ARE NEEDED TO DEFINE THE
NRSEL,Z,0               *  PATH FOR PATH OPERATIONS.
NRSEL,Y,22.5
NRSEL,X,0.3048
*GET,N45B,NDMN
NELEM                   * SELECT AGAIN ALL NODES ON STIF45 ELEMENTS
NRSEL,Z,0               * ALSO SELECT AND *GET THE NODE ON THE MID-PLANE
NRSEL,Y,22.5            *  OF THE FIN AT THE RADIUS OF THE PIPE O.D. (N45C)
NRSEL,X,0.1524          *  TO BE AN INTERMEDIATE PATH NODE.
*GET,N45C,NDMN
CSYS,0                  * ACTIVATE GLOBAL CARTESIAN CSYS
C*************************************************************************
C***       PATH DISPLAYS AND LINEARIZED STRESS DISPLAYS       ******
C*************************************************************************
ESEL,STIF,42            * FIRST WORK WITH STIF42 AXISYMMETRIC ELEMENTS
NELEM                   *     AND THE NODES ASSOCIATED WITH THE STIF42
/TITLE,PATH DISPLAYS FOR 2D AXISYMMETRIC MODEL
PLPATH,N42A,N42B,SX
PLPATH,N42A,N42B,SZ
/TITLE,LINEARIZED STRESS FOR 2D AXISYMMETRIC MODEL
PRSECT,N42A,N42B,-1
PLSECT,N42A,N42B,-1,SX
PLSECT,N42A,N42B,-1,SZ
C***   CSYS,0 IS ALSO USED FOR LINEARIZED STRESS CALCULATION
C***   ON THE 3D MODEL, BUT ONLY BECAUSE THE 3D MODEL WAS CONSTRUCTED
C***   SUCH THAT THE GLOBAL X-Y PLANE COINCIDES WITH THE SECTION PLANE
C***   USED FOR LINEARIZED STRESS CALCULATIONS.
ESEL,STIF,45            * NEXT WORK WITH STIF45 BRICK ELEMENTS
NELEM                   *  AND THE NODES ASSOCIATED WITH THE STIF45
/TITLE,PATH DISPLAYS FOR 3D SOLID MODEL
PLPATH,N45A,N45B,SX
PLPATH,N45A,N45B,SZ
/TITLE,LINEARIZED STRESS FOR 3D SOLID MODEL
PRSECT,N45A,N45B,-1
PLSECT,N45A,N45B,-1,SX
PLSECT,N45A,N45B,-1,SZ
C*************************************************************************
C***       GENERAL PATH OPERATIONS FOR LINEARIZED STRESS DISPLAYS      ******
C***       LINEARIZED STRESS COMPUTED ACCORDING TO THE EQUATIONS       ******
C***        IN CHAPTER 5, ANSYS THEORETICAL MANUAL, NOV 16,1987        ******
C***          EVALUATE EQUATIONS 5.2.25, 5.2.26, 5.2.27                ******
C***                             5.2.29, 5.2.30, 5.2.31                ******
C*************************************************************************
ESEL,STIF,42              * FIRST WORK WITH STIF42 AXISYMMETRIC ELEMENTS
```

```
NELEM                          *     AND THE NODES ASSOCIATED WITH THE STIF42
LPATH,N42A,N42C,N42B           * DEFINE PATH USING 3 NODES IN THE STIF42 MODEL
*GET,X1,NX,N42A                * GET THE X COORD. (RADIUS) OF FIRST PATH NODE
*GET,X2,NX,N42B                * GET THE X COORD. (RADIUS) OF LAST PATH NODE
RC=(X1+X2)/2                   * COMPUTE THE AVERAGE RADIUS OF THE SECTION
PDEF,INTR,SX                   * INTERPOLATE STRESS SX ON THE PATH
PDEF,INTR,SZ                   * INTERPOLATE STRESS SZ ON THE PATH
*GET,T1,PLAST,S                * GET THE PATH LENGTH = SECTION THICKNESS
C***   CREATE A PATH ITEM (XGC) WHICH IS CENTERED AT THE MIDDLE OF THE PATH
PCALC,ADD,XGC,XG,,,,-RC        * FOR USE IN INTEGRATION FROM -T/2 TO +T/2
C*************************************************************************
C***  COMPUTE LINEARIZED STRESS FROM THE SX STRESS COMPONENT ALONG THE PATH
C*************************************************************************
C***  INTEGRATE STRESS, SX, ALONG PATH = C1 = (1/T) * INTEGRAL OF SX dXGC
PCALC,INTG,C1,SX,XGC,(1/T1) *  THEORET. MANUAL EQUATION (5.2.25)
*GET,MEMX,PLAST,C1           * GET THE MEMBRANE STRESS IN THE SECTION FROM SX
*GET,SXB1,SX,N42A            * GET THE X-COMPONENT OF STRESS AT NODE 1
SXB1=SXB1-MEMX               * COMPUTE LIN. BEND. STRESS AS EQN. (5.2.26)
*GET,SXB2,SX,N42B            * GET THE X-COMPONENT OF STRESS AT NODE 2
SXB2=SXB2-MEMX               * COMPUTE LIN. BEND. STRESS AS EQN. (5.2.27)
C*************************************************************************
C***  COMPUTE LINEARIZED STRESS FROM THE SZ STRESS COMPONENT ALONG THE PATH
C*************************************************************************
C***  INTEGRATE STRESS, SZ, ALONG PATH = C1 = (1/T) * INTEGRAL OF SZ dXGC
C***    TO EVALUATE THE MEMBRANE LINEARIZED STRESS FROM THE HOOP STRESS, SZ
C*************************************************************************
PCALC,INTG,C1,SZ,XGC,(1/T1) *  THEORET. MANUAL EQUATION (5.2.29) IF RHO=INFINITY
*GET,MEMZ,PLAST,C1           * GET THE MEMBRANE STRESS IN THE SECTION FROM SZ
C*************************************************************************
C***  COMPUTE THE BENDING LIN. STRESS FOR THE HOOP STRESS, SZ
C*************************************************************************
C***  FIRST MULTIPLY STRESS, SZ, AND SECTION COORDINATE, XGC
PCALC,MULT,C1,SZ,XGC
C***  INTEGRATE THIS PRODUCT WRT THE SECTION COORDINATE, dXGC
PCALC,INTG,SBZ,C1,XGC
*GET,SBZ,PLAST,SBZ
X1=X1-RC                       * COMPUTE X1 RELATIVE TO CENTER OF SECTION
CON=(12*X1)/(T1**3)            * CON IS THE MULTIPLIER IN EQN. (5.2.30)
SBZ1=SBZ*CON                   * COMPUTE THE LIN. BENDING STRESS AT NODE 1 FROM SZ
X2=X2-RC                       * COMPUTE X2 RELATIVE TO CENTER OF SECTION
CON=(12*X2)/(T1**3)            * CON IS THE MULTIPLIER IN EQN. (5.2.31)
SBZ2=SBZ*CON                   * COMPUTE THE LIN. BENDING STRESS AT NODE 2 FROM SZ
*STAT                          * LIST ALL PARAMETERS
C*********************************************
C***    DISPLAY PATH IN STIF45 MODEL   ******
C*********************************************
ESEL,STIF,45                   * NEXT WORK WITH STIF45 BRICK ELEMENTS
NELEM                          *  AND THE NODES ASSOCIATED WITH THE STIF45
LPATH,N45A,N45C,N45B           * DEFINE PATH USING 3 NODES IN THE STIF45 MODEL
PDEF,INTR,SX                   * INTERPOLATE STRESS SX ON THE PATH
/TITLE,PATH DATA DISPLAY IN 3D MODEL
PVIEW,PLOT,SX                  * DISPLAY STRESS SX ALONG THE PATH
PVIEW,PRINT,SX                 * PRINT STRESS SX ALONG THE PATH
```

```
/VIEW,1,1,2,3                * SET THE VIEWPOINT FOR THE DISPLAY
/TITLE,DISPLAY OF PATH IN 3D MODEL
/EDGE,1,1                    * REMOVE INTERIOR ELEMENT OUTLINES
EPLOT                        * DISPLAY ELEMENTS
/EDGE,1,0                    * DEACTIVATE EDGE OPTION
/NOERASE                     * OVERLAY NEXT DISPLAY
/PBC,PATH,1                  * DISPLAY THE PATH GEOMETRY ON SUBSEQUENT DISPLAYS
NPLOT                        * DISPLAY THE NODES OF THE 3D STIF45 MODEL
FINISH
```

3.5 Vector Data

```
/GET,12,THERMAL,12,[DJ.TUT44.SECT1_EX]       * Attach file: THERMAL.12
/SHOW,VECT,F33,1             * Request vector graphics on file: VECT.F33
/POST1                       * Enter POST1
STRESS,THER                  * Store thermal centroid vector data
SET,1,1                      * Read data from postdata file, load step 1, iter. 1
ESEL,STIF,70                 * Select all STIF70 elements
NELEM                        * Select all nodes on STIF70 elements
/VIEW,1,1,3,1                * Set view for 3D model display
/EDGE,1,1                    * Remove interior element edges on displays
CSYS,11                      * Activate local coordinate system 11
NRSEL,Y,18,23                * Select nodes between theta of 18 and 23 degrees
ENODE,1                      * Select only elements with all nodes selected
/TITLE,THERMAL FLUX VECTOR DATA IN SLICE OF THE 3D MODEL
PLVECT,TF                    * Plot vectors of thermal flux
PRVECT,TF                    * Print a listing of thermal flux vectors
FINISH                       * Exit POST1
/GET,12,STRESS,12,[DJ.TUT44.SECT1_EX]          * Attach file: STRESS.12
/POST1                       * Enter POST1
SET,2,1                      * Read data from postdata file, load step 2, iter. 1
ESEL,STIF,45                 * Select all STIF45 elements
NELEM                        * Select all nodes on STIF45 elements
CSYS,11                      * Activate local coordinate system 11
NRSEL,Y,18,23                * Select nodes between theta of 18 and 23 degrees
ENODE,1                      * Select only elements with all nodes selected
/VIEW,1,1,3,1                * Set view for 3D model display
/EDGE,1,1                    * Remove interior element edges on displays
/TITLE,PRINCIPAL STRESS DATA IN SLICE OF THE 3D MODEL
PLVECT,PDIR                  * Produce principal stress vector display
PRVECT,PDIR                  * Print a listing of principal stress vector data
/TITLE,DISPLACEMENT VECTOR DATA IN SLICE OF THE 3D MODEL
PLVECT,DISP                  * Display displacement vectors
PRVECT,DISP                  * Print displacement vector data
FINISH                       * Exit POST1
/POST1                       * Enter POST1
STRESS,CSY,42,2              * Store data item 2 for STIF42 as CSY
STRESS,CGX,42,61             * Store data item 61 for STIF42 as CGX
STRESS,CGY,42,62             * Store data item 62 for STIF42 as CGY
SET,2,1                      * Read data from postdata file, load step 2, iter. 1
ESEL,STIF,42                 * Select all STIF42 elements
NELEM                        * Select all nodes on STIF42 elements
```

```
SADD,ZERO,CGX,CGX,,-1       * Create a labeled stress item with all values = zero
SADD,CGX,CGX,,-1            * Change the sign of the CGX data set
SADD,CGY,CGY,,-1            * Change the sign of the CGY data set
NRSEL,X,0,0.163             * Select nodes of the pipe and at the root of the fin
ENODE,1                     *  and the attached elements.
/VSCALE,1,2                 * Assign double scaling for display vectors.
/TITLE,POSITION VECTORS OF 2D MODEL ELEMENT CENTROIDS
C***   Create a vector display named LOC using CGX, CGY, and ZERO as the three
PLVECT,CGX,CGY,ZERO,LOC     * components to make into a vector
/VSCALE,1,0                 * Restore automatic length scaling for vectors
/TITLE,AXIAL STRESS COMPONENT VECTOR DISPLAY
C***   Create a vector display named SIGY using ZERO, CSY, and ZERO as the three
PLVECT,ZERO,CSY,ZERO,SIGY   * vector components to form the vector.
FINISH                      * Exit POST1
```

3.6 Line Element Processing

```
/SHOW,BEAM,F33,1        * Request vector graphics displays on file:BEAM.F33
/PREP7                  * Begin preprocessing
/TITLE, EXM-7C, STATIC ANAL. OF A SYSTEM OF INTERFACING BEAMS
/COM    This example is taken from the Substructures and Submodels seminar
ET,1,3                  * STIF3 two-dimensional beam element
ET,2,12                 * STIF12 two-dimensional interface
R,1,2,.1666,1           * Upper beam - area, moment of inertia, and thickness
R,2,4,1.3333,2          * Lower beam - area, moment of inertia, and thickness
R,3,0.0,100000,-.2      * Interface  - orientation, stiffness, and clearance
MP,EX,1,30E6            * Material properties of steel used for both beams.
MP,ALPX,1,8E-6
MP,DENS,1,.00073
N,1
N,11,100
FILL
NGEN,2,20,1,11,1,,-.01 * Generate a nearly coincident set of node points
E,1,2                   * Elements 1-10 are the upper beam
EGEN,10,1,1
EGEN,2,20,-10,,,,,1     * Generate the lower beam elements with RINC=1
TYPE,2
REAL,3
E,22,2                  * Define interface elements 21-29
EGEN,9,1,-1
ITER,-10,11,10          * Allow up to 10 iterations
ACEL,,386.4             * Define the gravity effect in the global Y direction
D,1,ALL,,,31,10         * Constrain both ends of both beams
TREF,70                 * Define the reference temperature for thermal loading
KTEMP,-1                * Temperature loads will be specified in PREP7
TE,11,70,100            * Use TE and TEGEN to define element temperatures
TEGEN,10,1,11           *    on elements 11-20 (lower beam).
F,2,FY,-186,,10         * Define FY forces on upper beam nodes.
WSORT,X
AFWRITE
FINISH
/INPUT,27
```

```
FINISH
/POST1
STRESS,FN,12,1          * Store  FN, the STIF12 normal force
STRESS,MZI,3,3          * Store MZI, the STIF3 moment at node I
STRESS,MZJ,3,6          * Store MZJ, the STIF3 moment at node J
STRESS,FYI,3,2          * Store FYI, the STIF3 Y force at node I
STRESS,FYJ,3,5          * Store FYJ, the STIF3 Y force at node J
STRESS,SBI,3,10         * Store SBI, the STIF3 bending stress at node I
STRESS,SBJ,3,14         * Store SBJ, the STIF3 bending stress at node J
SET,1,10                * Read postdata for the converged iteration.
SADD,MZJ,MZJ,,-1        * Change the sign of all J-node moment values
SADD,FYJ,FYJ,,-1        * Change the sign of all J-node Y force values
ESEL,REAL,1             * Select upper beam elements.
/TITLE, BENDING MOMENT IN UPPER BEAM
PLLS,MZI,MZJ            * Display bending moment variation along upper beam.
/TITLE, SHEAR FORCE IN UPPER BEAM
PLLS,FYI,FYJ            * Display Y force variation along upper beam.
/TITLE, BENDING STRESS IN UPPER BEAM
PLLS,SBI,SBJ            * Display bending stress variation along upper beam.
ESEL,REAL,2             * Select lower beam elements.
/TITLE, BENDING MOMENT IN LOWER BEAM
PLLS,MZI,MZJ            * Display bending moment variation along lower beam.
/TITLE, SHEAR FORCE IN LOWER BEAM
PLLS,FYI,FYJ            * Display Y force variation along lower beam.
/TITLE, BENDING STRESS IN LOWER BEAM
PLLS,SBI,SBJ            * Display bending stress variation along lower beam.
ESEL,TYPE,2             * Select all interface elemenets
/TITLE,NORMAL FORCE IN INTERFACE ELEMENTS
/PNUM,SVAL,1            * Request stress value numbering on plots.
PLNSTR,FN               * Display result FN on selected elements.
FINISH
```

3.7 Dump Truck Modal Analysis

```
/SHOW,DUMPK2,F33          * Request displays on plot file: DUMPK2.F33
/PREP7
/TITLE, DUMP TRUCK - MODAL ANALYSIS
KAN,2                     * Analysis type set to 2
KAY,2,5                   * Expand 5 modes for postprocessing
ET,1,63                   * Use STIF63 thin shell elements
ET,2,63                   *   with two element types
R,1,.5                    * Thickness of center of lip overhanging truck cab.
R,2,1                     * Thickness of the sides of the dump truck bed.
R,3,1.5                   * Thickness of edge of overhanging lip.
MP,EX,1,3E7               * Material properties of steel.
MP,DENS,1,.00073          *
MP,NUXY,1,.3              *
K,1                       * Define geometry with solid modeling.
K,2,,,200
K,3,,,214
K,4,,,256
K,5,,,270
```

```
KGEN,2,1,5,1,120
K,11,,-20,-10
K,12,,-60,170
KGEN,2,11,12,1,120
KGEN,2,2,5,1,12
KGEN,2,2,5,1,108
L,2,5,5                    * Define mesh divisions for quad. meshing
L,16,17,3
L,15,19,8
RP4,1,1
L,15,16,1
L,17,18,1
L,5,18,1
L,22,10,1
L,7,10,5
L,15,2,1
L,22,21,1
L,20,19,1
L,7,19,1
L,20,21,3
L,18,15,5
L,19,22,5
A,2,5,18,15                * Areas 1-5 are lip overhanging the truck cab
A,18,22,21,17
A,16,20,19,15
A,22,10,7,19
A,17,21,20,16
TYPE,2
REAL,3
AMESH,1,4
REAL,1
AMESH,5
L,1,11,3
L,2,12,3
L,6,13,3
L,7,14,3
TYPE,1
REAL,1
ELSIZE,,10
A,12,14,7,2                * Areas 6 and 7 are the front and bottom of the
A,12,14,13,11              *   dump truck bed.
AMESH,6,7
REAL,2
A,14,13,6,7                * Areas 8 and 9 are the sides of the dump truck bed.
A,12,11,1,2
AMESH,8,9
/TYPE,1,4                  * Request precise hidden displays
/VIEW,1,1,1,-1             * Set viewpoint
/PNUM,REAL,1               * Numbering will be determined by REAL constant values
/TITLE, DUMP TRUCK - MODAL ANALYSIS - ELEMENTS
EPLOT                      * Display elements.
NSEL,NODE,75,85
NASEL,NODE,117,226
```

```
D,ALL,UY                    * Constrain nodes in truck bed in Y direction.
NALL
D,81,UX,,,,,UZ              * Constrain two other points in truck bed in X and Z
D,132,UX,,,,,uz             *    directions to prevent rigid body motion.
MERGE                       * Merge coincident nodes (in overhanging lip)
WSORT,Z                     * Reorder for solution efficiency.
M,1,UY,74                   * Pick MDOF on overhanging lip in Y direction.
TOTAL,100,1                 * Total of 100 MDOF ( automatic selection of 34 MDOF)
SVTYPE,2                    * Define a dummy acceleration spectrum to force the
SED,0,1,0                   *  calculation of stresses and displacements
FREQ,0.1,1000               *  because force data is needed in POST1 to display
SV,0.01,386.4,386.4         *  all MDOF using force vectors.
EXMODE,,,1E-8               * Expand modes of low significance
/PBC,ALL,1                  *
/TYPE,1,0                   * Normal, see-through display.
/TITLE, DUMP TRUCK NODES WITH MDOF AND DISPLACEMENT B.C.
NPLOT                       * Display nodes
AFWRIT
FINISH
/INPUT,27
FINISH
/POST1
/VIEW,1,1,1,-1
SET,1,1                     * Store results for the first mode of vibration.
/TITLE, DUMP TRUCK MODAL ANALYSIS - FIRST FOUR MODE SHAPES
/WIND,1,LTOP                * Use left-top 1/4 window for mode shape display.
/TYPE,1,4                   * Request precise hidden display.
PLDISP,1                    * Plot deformed shape, overlay undeformed geometry.
/NOERASE                    * Overlay next displays.
SET,1,2                     * Store results for the second mode of vibration.
/WIND,1,RTOP                * Use right-top 1/4 window for mode shape display.
PLDISP,1                    * Plot deformed shape, overlay undeformed geometry.
SET,1,3                     * Store results for the third mode of vibration.
/WIND,1,LBOT                * Use left-bottom 1/4 window for mode shape display.
PLDISP,1                    * Plot deformed shape, overlay undeformed geometry.
SET,1,4                     * Store results for the fourth mode of vibration.
/WIND,1,RBOT                * Use right-bottom 1/4 window for mode shape display.
PLDISP,1                    * Plot deformed shape, overlay undeformed geometry.
/ERASE                      * Erase screen between subsequent displays.
SET,1,1                     * Store results for the first mode of vibration.
/WIND,1,FULL                * Use the full display window for remaining displays.
/TYPE,1,0                   * Normal, see-through display.
/PBC,FORC,2                 * Display force vectors with "unity" scaling.
/PBC,TDIS,1                 * Display displacement boundary conditions.
NUSEL,X,1,119               * Display model in sections for better visualization.
/TITLE, DISPLAY OF DUMP TRUCK SIDE WALLS WITH B.C.
EPLOT                       * Only elements with nodes in the nodal subset can be
NSEL,,1,226                 *  displayed, though all elements are selected
/TITLE, DISPLAY OF DUMP TRUCK BED AND LIP WITH B.C.
EPLOT
/PBC,FORC,1                 * Display force vectors with "magnitude" scaling
EPLOT                       *  to show the value of the "unity" scaling option.
FINISH
```

3.8 Interfacing Beams – Static Analysys

```
/SHOW,BEAM2,F33          * Request raster displays on file: BEAM2.F33
/PREP7                   * Begin preprocessing
/TITLE, EXM-7C, STATIC ANAL. OF A SYSTEM OF INTERFACING BEAMS
/COM    This example is taken from the Substructures and Submodels seminar
ET,1,3                   * STIF3 two-dimensional beam element
ET,2,12                  * STIF12 two-dimensional interface
R,1,2,.1666,1            * Upper beam - area, moment of inertia, and thickness
R,2,4,1.3333,2           * Lower beam - area, moment of inertia, and thickness
R,3,0.0,100000,-.2       * Interface  - orientation, stiffness, and clearance
MP,EX,1,30E6             * Material properties of steel used for both beams.
MP,ALPX,1,8E-6
MP,DENS,1,.00073
N,1
N,11,100
FILL
NGEN,2,20,1,11,1,,-.01 * Generate a nearly coincident set of node points
E,1,2                    * Elements 1-10 are the upper beam
EGEN,10,1,1
EGEN,2,20,-10,,,,,1      * Generate the lower beam elements with RINC=1
TYPE,2
REAL,3
E,22,2                   * Define interface elements 21-29
EGEN,9,1,-1
ITER,-10,11,10           * Allow up to 10 iterations
ACEL,,386.4              * Define the gravity effect in the global Y direction
D,1,ALL,,,31,10          * Constrain both ends of both beams
TREF,70                  * Define the reference temperature for thermal loading
KTEMP,-1                 * Temperature loads will be specified in PREP7
TE,11,70,100             * Use TE and TEGEN to define element temperatures
TEGEN,10,1,11            *    on elements 11-20 (lower beam).
F,2,FY,-186,,10          * Define FY forces on upper beam nodes.
WSORT,X                  * Reorder
LWRITE                   * Load step 1 - uses nodal FY on upper beam
FDELE,2,FY,10,1          * Delete FY forces on upper beam nodes
EP,1,1,16.74,,10,1       * Define equivalent upper beam load using EP
LWRITE                   * Load step 2 - will have the same total load, but the
AFWRITE                  *  bending moment (and thus) the displacement results
FINISH                   *  will be slightly different (concentrated nodal loading
/INPUT,27                *  vs. uniformly distributed loading).
FINISH
/POST1
LCLIM,3                  * Allow up to 3 data storage arrays.
STRESS,FN,12,1           * Store  FN, the STIF12 normal force.
STRESS,MZI,3,3           * Store MZI, the STIF3 bending moment at node I.
STRESS,MZJ,3,6           * Store MZJ, the STIF3 bending moment at node J.
SET,1,10                 * Read postdata for the converged iteration, load step 1.
SADD,MZJ,MZJ,,-1         * Change the sign of th MZJ element data in load case 1.
LCASE,2                  * Set load case pointer to 2.
SET,2,10                 * Read postdata for the converged iteration, load step 2.
SADD,MZJ,MZJ,,-1         * Change the sign of th MZJ element data in load case 2.
ESEL,REAL,1              * Select elements of upper beam.
```

```
NELEM                  * Select nodes corresponding to selected elements.
/WINDOW,1,TOP          * Set graphics window to top 1/2 of frame.
/CONTOUR,1,5           * Request only 5 automatic contour levels.
/TITLE, BENDING MOMENT COMPARISON
LCASE,1                * Set load case pointer to 1.
PLLS,MZI,MZJ           * Display bending moment results in upper beam.
/WINDOW,1,BOTT         * Set graphics window to bottom 1/2 of screen.
LCASE,2                * Set load case pointer to 2.
/NOERASE               * Overlay displays.
PLLS,MZI,MZJ           * Display bending moment results in upper beam.
/ERASE                 * Clear display between plot requests.
/WINDOW,1,TOP          * Set graphics window to top 1/2 of frame.
/DSCALE,1,75           * Set distortion scale factor to 75.
/TITLE, DEFORMED SHAPE COMPARISON
LCASE,1                * Set load case pointer to 1.
PLDISP,1               * Display deformed geometry, overlay undeformed shape.
LCASE,2                * Set load case pointer to 2.
/NOERASE               * Overlay displays.
PLDISP,1               * Display deformed geometry, overlay undeformed shape.
/ERASE                 * Clear display between plot requests.
/DSCALE,1,AUTO         * Use automatic distortion scaling.
LCFACT,1,-1            * Define load case factors, 1 and -1.
LCADD,3,1,2            * Create load case 3 = load case 1 - load case 2.
LCASE,3                * Set load case pointer to 3.
/TITLE, BENDING MOMENT AND DEFORMED SHAPE DIFFERENCES
PLLS,MZI,MZJ           * Display bending moment results in upper beam.
/DSCALE,1,1000         * Set distortion scale factor to 1000.
/WINDOW,1,BOTT         * Set graphics window to bottom 1/2 of screen.
/NOERASE               * Overlay displays.
PLDISP,1               * Display deformed geometry, overlay undeformed shape.
/ERASE                 * Clear display between plot requests.
/TITLE, RESULTS FOR CONCENTRATED NODAL FORCE LOADING
LCASE,1                * Set load case pointer to 1.
ESEL,TYPE,2            * Select all STIF12 interface elements.
PRSTRS                 * Print element data (FN) for selected elements.
ESEL,REAL,1            * Select elements of the upper beam
NELEM                  * Select all nodes on the set of selected elements
NFORCE                 * Print forces and force summation for selected sets.
/TITLE, RESULTS FOR UNIFORMLY DISTRIBUTED PRESSURE LOADING
LCASE,2                * Set load case pointer to 1.
ESEL,TYPE,2            * Select all STIF12 interface elements.
PRSTRS                 * Print element data (FN) for selected elements.
ESEL,REAL,1            * Select elements of the upper beam
NELEM                  * Select all nodes on the set of selected elements
NFORCE                 * Print forces and force summation for selected sets.
FINISH
/SHOW,BEAM2V,F33,1     * Request vector displays on file: BEAM2.F33
/POST1
LCLIM,3                * Allow up to 3 data storage arrays.
STRESS,FN,12,1         * Store  FN, the STIF12 normal force.
STRESS,MZI,3,3         * Store MZI, the STIF3 bending moment at node I.
STRESS,MZJ,3,6         * Store MZJ, the STIF3 bending moment at node J.
SET,1,10               * Read postdata for the converged iteration, load step 1.
```

```
SADD,MZJ,MZJ,,-1        * Change the sign of th MZJ element data in load case 1.
LCASE,2                 * Set load case pointer to 2.
SET,2,10                * Read postdata for the converged iteration, load step 2.
SADD,MZJ,MZJ,,-1        * Change the sign of th MZJ element data in load case 2.
ESEL,REAL,1             * Select elements of upper beam.
NELEM                   * Select nodes corresponding to selected elements.
/WINDOW,1,TOP           * Set graphics window to top 1/2 of frame.
/CONTOUR,1,5            * Request only 5 automatic contour levels.
/TITLE, BENDING MOMENT COMPARISON
LCASE,1                 * Set load case pointer to 1.
PLLS,MZI,MZJ            * Display bending moment results in upper beam.
/WINDOW,1,BOTT          * Set graphics window to bottom 1/2 of screen.
LCASE,2                 * Set load case pointer to 2.
/NOERASE                * Overlay displays.
PLLS,MZI,MZJ            * Display bending moment results in upper beam.
/ERASE                  * Clear display between plot requests.
/WINDOW,1,TOP           * Set graphics window to top 1/2 of frame.
/DSCALE,1,75            * Set distortion scale factor to 75.
/TITLE, DEFORMED SHAPE COMPARISON
LCASE,1                 * Set load case pointer to 1.
PLDISP,1                * Display deformed geometry, overlay undeformed shape.
LCASE,2                 * Set load case pointer to 2.
/NOERASE                * Overlay displays.
PLDISP,1                * Display deformed geometry, overlay undeformed shape.
/ERASE                  * Clear display between plot requests.
/DSCALE,1,AUTO          * Use automatic distortion scaling.
LCFACT,1,-1             * Define load case factors, 1 and -1.
LCADD,3,1,2             * Create load case 3 = load case 1 - load case 2.
LCASE,3                 * Set load case pointer to 3.
/TITLE, BENDING MOMENT AND DEFORMED SHAPE DIFFERENCES
PLLS,MZI,MZJ            * Display bending moment results in upper beam.
/DSCALE,1,1000          * Set distortion scale factor to 1000.
/WINDOW,1,BOTT          * Set graphics window to bottom 1/2 of screen.
/NOERASE                * Overlay displays.
PLDISP,1                * Display deformed geometry, overlay undeformed shape.
/ERASE                  * Clear display between plot requests.
FINISH
```

3.9 Cantilever Beam with Distributed Load

```
/PREP7
/TITLE,ANALYSIS OF CANTILEVER BEAM WITH DISTRIBUTED LOAD
ET,1,45                 * Use STIF45 3D solid elements.
MP,EX,1,3E7             * Young's modulus of steel in psi.
N,1                     * Define model by direct definition of nodes and
N,5,,2                  *  and elements.
FILL                    *
NGEN,3,5,1,5,1,,,0.5    * The beam ranges from 0 to 20 in length (x coordinate)
NGEN,11,15,1,15,1,2.0   *  from 0 to 2 in height (y coordinate) and from
E,1,2,7,6,16,17,22,21   *  0 to 1 in depth (z coordinate).
EGEN,2,5,-1             * The centerline of the beam runs from (0,1,0.5) to
EGEN,4,1,-2             *  (20,1,0.5).
```

```
EGEN,10,15,-8              *
D,1,UX,,,15,1              * The left end is constrained in all directions.
D,1,UZ,,,5                 *
D,1,UY,,,11,5              *
PSF,,2,2,10                * Pressure of 10 psi is applied on the top of the beam
AFWRIT                     *
FINISH
/INPUT,27
FINISH
/POST1
SET
SPOINT,,0,1,0.5
FSUM
/TITLE,FORCES DISPLAYED AT EACH SECTION ALONG THE BEAM LENGTH
*CREATE,MAC
NSEL,X,-0.01,ARG1+0.01       * Select all nodes and elements from the left end
ENODE,1                      *  of the beam up to x = ARG1.
NSEL,X,ARG1-0.01,ARG1+0.01   * Select only the nodes on the cut face (x = ARG1).
SPOINT,,ARG1,1,0.5           * Locate a point in 3D space for moment summation
NFORCE                       * Print and sum nodal forces.
*END
*USE,MAC,4                   * Sum at several node planes,starting from left end
RP5,,4
FINISH
```

3.10 Symmetric Analysis – Cantilever Beam

```
/PREP7
/TITLE,SYMMETRIC ANALYSIS OF CANTILEVER BEAM WITH DISTRIBUTED LOAD
ET,1,45                    * Use STIF45 3D solid elements.
MP,EX,1,3E7                * Young's modulus of steel in psi.
N,1                        * Define model by direct definition of nodes and
N,5,,2                     *  and elements.
FILL                       *
NGEN,2,5,1,5,1,,,0.5       * The beam ranges from 0 to 20 in length (x coordinate)
NGEN,11,15,1,10,1,2.0      *  from 0 to 2 in height (y coordinate) and from
E,1,2,7,6,16,17,22,21      *  0 to 0.5 in depth (z coordinate).
EGEN,4,1,-1                * The centerline of the beam runs from (0,1,0) to
EGEN,10,15,-4              *  (20,1,0).
D,1,UX,,,10,1              * The left end is constrained in all directions.
D,1,UY,,,6,5               *
SYMBC,0,3,0                * The global XY plane is a symmetry plane.
PSF,,2,2,10                * Pressure of 10 psi is applied on the top of the beam
AFWRIT                     *
FINISH
/INPUT,27
FINISH
/POST1
SET
SPOINT,,0,1,0
FSUM
/TITLE,FORCES DISPLAYED AT EACH SECTION ALONG THE BEAM LENGTH
```

```
*CREATE,MAC
NSEL,X,-0.01,ARG1+0.01        * Select all nodes and elements from the left end
ENODE,1                       *  of the beam up to x = ARG1.
NSEL,X,ARG1-0.01,ARG1+0.01    * Select only the nodes on the cut face (x = ARG1).
SPOINT,,ARG1,1,0              * Locate a point in 3D space for moment summation
NFORCE                        * Print and sum nodal forces.
*END
*USE,MAC,4                    * Sum at several node planes,starting from left end
RP5,,4
FINISH
```

3.11 Error Norm Example – Elliptic Membrane

```
/SHOW,PROB1,F33
TYP=42                          * Parameter for element type choice.
MDEN=4                          * Parameter for mesh density.
/PREP7
/TITLE, ELLIPTIC MEMBRANE
ET,1,TYP,,,3
ELSIZE,,MDEN
MP,EX,1,210E9                   * Young's modulus in Pa.
MP,NUXY,1,0.3                   * Poisson's ratio.
R,1,0.1                         * Define element thickness.
/COM,     Create model with six areas to comply with benchmark
/COM,     problem requirements.
LOCAL,11,1,,,,,,,0.5            * Define a local elliptical coordinate system.
K,1,2,90                        * Define keypoints on inside boundary.
K,4,2,0
K,5,1.165,20
KMOVE,5,0,1.165,999,0.0,11,2.0,999,0.0
K,8,2.0,5.0
KMOVE,8,0,999,0.453,0.0,11,2.0,999,0.0
L,1,5                           * Define line segments on inside boundary.
L,5,8
L,8,4
LOCAL,12,1,,,,,,,0.8461585      * Define a local elliptical coordinate system.
K,2,3.25,90                     * Define keypoints on outside boundary.
K,3,3.25,0.0
K,6,3.25,67
KMOVE,6,0,1.783,999,0.0,12,3.25,999,0.0
K,7,3.25,25
KMOVE,7,0,999,1.348,0.0,12,3.25,999,0.0
L,2,6                           * Define line segments on outside boundary.
L,6,7
L,7,3
LOCAL,13,1,,,,,,,0.65494415     * Define a local elliptical coordinate system.
K,9,2.417                       * Define keypoint for intermediate boundary.
K,10,2.417,90
L,9,10                          * Define line segment for intermediate boundary.
CSYS,0                          * Activate global cartesian coordinate system.
L,5,6                           * Define line segments from inside to outside.
L,7,8
```

```
LINTER,7,8                    * Intersect intermediate boundary line with both
LINTER,7,9                    *  inside to outside line segments
A,1,5,11,10                   * Define areas.
A,10,11,6,2
A,11,12,7,6
A,5,8,12,11
A,8,4,9,12
A,12,9,3,7
AMESH,ALL                     * Create nodes and elements.
CSYS,13
WSORT,Y                       * Reorder for solution efficiency.
CSYS,0
*GET,MAXN,NMAX                * Get the maximum node number of the model.
NDOF=MAXN*2                   * Compute the total number of dofs.
LSBC,14                       * Define symmetry boundary conditions on the
RP4,1                         *   XZ and YZ global cartesian planes.
LPSF,4,10E6                   * Define external pressure on outside boundary.
RP3,1
KSE,1                         * Request strain energy calculation.
SFWRIT
/PNUM,TYPE,1
/NUM,1
APLOT                         * Display geometry with an area plot.
/PBC,TDIS,1
/PBC,PRESS,2
EPLOT                         * Check boundary conditions on an element plot.
/SHOW,PROB1,F33,1             * Same display in vector graphics.
EPLOT
FINISH
/SOLVE
FINISH
/POST1
STRESS,ENRM                   * Request storage of labelled stress items:
STRESS,SENE                   *  ENRM, VOLU, and SENE (strain energy).
SET,1,1
NSEL,Y,0,.01                  * Request only the inner corner node.
NRSEL,X,2.0
*GET,NN,NDMX                  * Get the node number of the inner corner node.
*GET,DISP,UX,NN               * Get the UX displacement here.
*GET,SIGY,SY,NN               * Get the SY stress here.
SSUM                          * Sum the data in the labelled stress tables.
*GET,ENR,SSUM,ENRM            * Get the total error energy.
*GET,TSE,SSUM,SENE            * Get the total strain energy.
ERR=(ENR/(TSE+ENR))           * Calaculate the energy norm.
ERR=SQRT(ERR)                 * Calculate the error energy ratio.
ERR=ERR*100                   * Present the error energy ratio as a % error.
NN=608.3717                   * Total strain energy from very fine STIF82 model
TSER=((TSE-NN)/NN)*100        * Compute % error in total strain energy.
NN=-92.7E6                    * Theoretical stress at inside corner node.
SERR=((SIGY-NN)/NN)*100       * Compute % error in stress at inside corner.
NN=0.1021975E-3               * Radial displacement from very fine STIF82 model
DERR=((DISP-NN)/NN)*100       * Compute % error in radial displacement.
*STAT                         * List all parameter values.
```

```
FINISH
```

3.12 Adaptive Meshing of Elliptic Membrane

```
TYP=2                          * Define the element type, STIF2, 6 node triangle.
TARG=5                         * Initialize max. acceptable error to 5% (target).
LOPC=0                         * Initialize loop counter to zero.
KP=0                           * Initialize point of max. error to zero.
/PREP7
/TITLE, ELLIPTIC MEMBRANE
LOPC=LOPC+1                    * Increment loop counter by one each loop.
*IF,LOPC,NE,1,:NFS             * Bypass model definition for adaptive loops.
ET,1,TYP,,,3
MP,EX,1,210E9                  * Young's modulus in Pa.
MP,NUXY,1,0.3                  * Poisson's ratio.
R,1,0.1                        * Define element thickness.
/COM,     Create model with six areas to comply with benchmark
/COM,     problem requirements.
LOCAL,11,1,,,,,,,0.5           * Define a local elliptical coordinate system.
K,1,2,90                       * Define keypoints on inside boundary.
K,4,2,0
K,5,1.165,20
KMOVE,5,0,1.165,999,0.0,11,2.0,999,0.0
K,8,2.0,5.0
KMOVE,8,0,999,0.453,0.0,11,2.0,999,0.0
L,1,5                          * Define line segments on inside boundary.
L,5,8
L,8,4
LOCAL,12,1,,,,,,,0.8461585     * Define a local elliptical coordinate system.
K,2,3.25,90                    * Define keypoints on outside boundary.
K,3,3.25,0.0
K,6,3.25,67
KMOVE,6,0,1.783,999,0.0,12,3.25,999,0.0
K,7,3.25,25
KMOVE,7,0,999,1.348,0.0,12,3.25,999,0.0
L,2,6                          * Define line segments on outside boundary.
L,6,7
L,7,3
LOCAL,13,1,,,,,,,0.65494415    * Define a local elliptical coordinate system.
K,9,2.417                      * Define keypoint for intermediate boundary.
K,10,2.417,90
L,9,10                         * Define line segment for intermediate boundary.
CSYS,0                         * Activate global cartesian coordinate system.
L,5,6                          * Define line segments from inside to outside.
L,7,8
LINTER,7,8                     * Intersect intermediate boundary line with both
LINTER,7,9                     *  inside to outside line segments
A,1,5,11,10                    * Define areas.
A,10,11,6,2
A,11,12,7,6
A,5,8,12,11
A,8,4,9,12
```

```
A,12,9,3,7
CSYS,12                 * Acivate elliptical coordinate system 12.
KPSEL,X,3.25            * Select keypoints on outside of model.
KDVS,ALL,1.0            * Element edge lengths at these keypoints = 1.0
CSYS,13                 * Acivate elliptical coordinate system 13.
KPSEL,X,2.417           * Select keypoints on intermediate boundary.
KDVS,ALL,0.75           * Element edge lengths at these keypoints = 0.75
CSYS,11                 * Acivate elliptical coordinate system 11.
KPSEL,X,2.0             * Select keypoints on inside of model.
KDVS,ALL,0.5            * Element edge lengths at these keypoints = 0.5
KPALL                   * All keypoints are active.
/SHOW,PROB2,F33,1       * Define plot data filename - vector graphics.
/PNUM,POIN,1            * Include keypoint numbers on solid model displays.
LPLOT                   * Display lines.
/PNUM,POIN,0            * Deactivate keypoint display key.
/PNUM,LINE,1            * Include line numbers on solid model displays.
LPLOT                   * Display lines.
/PNUM,TYPE,1            * Include element type numbering on displays.
/NUM,1                  * Use color to distinguish diff. numered items.
*CREATE,PMAC            * Put display commands in a macro for looping.
/SHOW,PROB2,F33         * Define plot data filename
EPLOT                   * Display elements.
/SHOW,PROB2,F33,1       * Same display in vector graphics.
EPLOT
*END                    * End macro definition.
*GO,:MSH
:NFS
RESUME,,,,1             * Restore previous model, retain current parameters.
*GET,KP,KPNE,EEMX       * Get the point nearest the elem. with max. error.
PDEL,ALL,ALL            * Delete all loads from previous loop.
DDEL,ALL,ALL            * Delete all boundary conditions from prev. loop.
ACLEAR,ALL              * Clear previous model.
NUMCMP,ALL              * Compress all numbered items.
KPSEL,POIN,KP           * Select the keypoint nearest the max. error.
LSKP                    * Select the lines and areas which attach to that
ARLS                    *  point.  Then select all lines and keypoints near
LSAR                    *  the max. error location and adjust the mesh in
KPLS                    *  only that region.
KDVS,ALL,,0.9           * Increase mesh density 10% at selected keypoints.
KDVS,KP,,0.6            * Increase mesh density an additional 40% at the
:MSH                    *      keypoint nearest the maximum error.
KPALL                   * Activate all keypoints.
LSALL                   * Activate all lines.
ARALL                   * Activate all areas.
LDVA                    * Adjust divisions and spacing ratio on all lines.
AMESH,ALL               * Create nodes and elements.
*GET,MAXN,NMAX          * Get the maximum node number of the model.
NDOF=MAXN*2             * Compute the total number of dofs.
WAVES                   * Reorder the mesh.
LSBC,14                 * Define symmetry boundary conditions on the
RP4,1                   *   XZ and YZ global cartesian planes.
LPSF,4,10E6             * Define external pressure on outside boundary.
RP3,1
```

```
KSE,1                          * Request strain energy calculation.
SFWRIT
*USE,PMAC                      * Produce displays using macro.
FINISH
/SOLVE
FINISH
/POST1
STRESS,ENRM                    * Request storage of labelled stress items:
STRESS,SENE                    *  ENRM, VOLU, and SENE (strain energy).
SET,1,1
NSEL,Y,0,.001                  * Request only the inner corner node.
NRSEL,X,2.0
*GET,NN,NDMX                   * Get the node number of the inner corner node.
*GET,DISP,UX,NN                * Get the UX displacement here.
*GET,SIGY,SY,NN                * Get the SY stress here.
SSUM                           * Sum the data in the labelled stress tables.
*GET,ENR,SSUM,ENRM             * Get the total error energy.
*GET,TSE,SSUM,SENE             * Get the total strain energy.
ERR=(ENR/(TSE+ENR))            * Calaculate the energy norm.
ERR=SQRT(ERR)                  * Calculate the error energy ratio.
ERR=ERR*100                    * Present the error energy ratio as a % error.
NN=608.3717                    * Total strain energy from very fine STIF82 model
TSER=((TSE-NN)/NN)*100         * Compute % error in total strain energy.
NN=-92.7E6                     * Theoretical stress at inside corner node.
SERR=((SIGY-NN)/NN)*100        * Compute % error in stress at inside corner.
NN=0.1021975E-3                * Radial displacement from very fine STIF82 model
DERR=((DISP-NN)/NN)*100        * Compute % error in radial displacement.
*STAT                          * List all parameter values.
ESORT,ENRM,,1,1                * Sort elements based on ENRM value.
*GET,ERMX,MAX                  * Get the value of the maximum error norm.
ESEL,ENRM,ERMX                 * Select only the element with the max. error.
*GET,EEMX,ELMX                 * Get element number of element with max. error.
FINISH
/OPT
OPOUT,24                       * Divert output during looping.
OPVAR,KP,DV,0,99               * Declare the model parameters as dummy variables
OPVAR,NDOF,DV,0,4000           *  in the optimization routine. The choice of
OPVAR,ERR,SV,-100,100          *  variable type (DV, SV, OBJ) and the variable
OPVAR,TSER,SV,-100,100         *  limits is arbitrary since optimization curve
OPVAR,SERR,SV,-100,100         *  fitting (approximations) are not done (e.g. the
OPVAR,DERR,OBJ                 *  OPRUN command is not used.). Only these
OPCOPY                         *  are printed at OPLIST ( for a concise summary).
:RRN                           *
RUN                            * Loop on the RUN command if the error exceeds
*IF,ERR,GT,TARG,:RRN           *  the target.
OPLIST,ALL                     * List the summary of the optimization variables.
/SHOW,PROB2,F33,1              * Display in vector graphics.
OPSORT,NDOF                    * Sort data sets into ascending NDOF order.
XVAR,NDOF                      * Use NDOF as the abcissa for displays.
/GRAPH,LOGX,1                  * Use log scale on X-axis.
/GRAPH,LOGY,1                  * Use log scale on Y-axis.
/TITLE, ERROR NORM VARIATION WITH NDOF
PLVAR,ERR                      * Display the error norm variation.
```

```
FINISH
```

4. Nonlinear Dynamics

4.1 Nonlinear Impulsively Loaded Clamped Plate

```
/PREP7
/TITLE, NONLINEAR IMPULSIVELY LOADED CLAMPED PLATE
C*** From "Explicit Algorithms For Nonlinear Dynamics of Shells,"
C***   T.B. Belytschko and C.S. Tsay, NONLINEAR FINITE ELEMENT ANALYSIS OF
C***   PLATES AND SHELLS, T.J.R. Hughes, et. al., Ed., AMD-48, ASME, P209-231, 1981.
C*** Define analysis options
KAN,4
KAY,5,0                                 * Define initial velocity
KAY,6,1                                 * Large deflection
GAMMA,,8                                * Up the no. of equilibrium iterations
ET,1,43
R,1,.125
C*** Define material properties
MP,EX,,10.4E6
MP,NUXY,,.3
MP,DENS,,2.61E-4
KNL,1
NL,,13,10                               * C13=10, Bilinear Kinematic Hardening
NL,,25,41400                            * Yield=41400, Tangent modulus=0
C*** Define geometry - model one quarter of the plate
N,1
N,11,5
FILL
NGEN,2,20,1,11,1,,.6
E,1,2,22,21
EGEN,10,1,1
C*** Impose initial velocity = 5000 over central section only
ITER,1,0,1
TIME,1E-8
D,ALL,ALL
NSEL,X,0,1
D,ALL,UZ,-5E-5
NALL
LWRI
C*** Define remaining transient
ITER,100,0,1
TIME,0.001
DDEL,ALL,ALL
C*** Define boundary conditions
SYMBC,,1
SYMBC,,2
NSEL,X,5
D,ALL,ALL                               * Fixed at end
NALL
LWRI
AFWR
FINI
/INP,27
```

```
FINI
/POST26
C*** Retrieve deflection history
DISP,2,1,UZ
ADD,3,2,,,DISP,,,-1.0                    * Reverse sign on UZ
PRVA,3
/GRAPH,LABY,DISP
PLVA,3
FINI
```

4.2 Pipe Whip Analyssis Including Restraint

```
/show,pw,f33
/prep7
/title, Pipe Whip Analysis Including Restraint
c*** From "On Some Current Procedures and Difficulties in Finite Element
c***    Analysis of Elastic-Plastic Response," K.J. Bathe et. al., COMPUTERS
c***    AND STRUCTURES, Vol. 12, No. 4, 1980, p607-624.
c*** Define pipe properties
et,1,20
r,1, 30,1.125                            * pipe diameter and wall thickness
mp,ex,,2.698e7
mp,nuxy,,.3
mp,dens,,7.18e-4
knl,1
nltab,,2                                 * bilinear stress-strain behavior
nly,define,,,29140,0                     * yield stress, tangent slope
c*** Define restraint properties
et,2,40,,,2,,1                           * UY restraint, 5% converg. tolerance
A=(3.14159*(5.75**2))/4                  * cross sectional area
E=2.99e7                                 * modulus of elasticity
L=21                                     * length
SIGY=38000                               * yield stress
r,2, (A*E)/L,,,3,A*SIGY                  * stiffness, gap and fslide
c*** Set up geometry
n,1
n,2,6                                    * small 1st ele to capture plast hinge
n,8,360
fill
e,1,2
egen,7,1,1
type,2
real,2
n,9,360
e,9,8                                    * node order defines gap, not hook
c*** Define constraints
d,1,all
d,9,uy
d,all,uz                                 * prevent out-of-plane motion
c*** Plot model with BC and node numbers
/pbc,all,1
/pnum,node,1
```

```
eplot
c***
c*** First perform modal analysis
c***   Note that the gap will be ignored since it is initially open
c***
kan,2
kay,1,-1                                    * full subspace (no MDOF needed)
kay,7,2                                     * obtain first 2 modes
kay,2,2                                     * put 2 modes on postdata file
afwr
fini
/inp,27
fini
c*** Look at 1st two modes
/post1
set,1,1
pldi,1
set,1,2
pldi,2
fini
c***
c*** Now perform dynamic analysis
c***
/prep7
resu
kan,4
kay,1
kay,7
kay,5,2                                     * start at rest
c*** time step selected to resolve restraint frequency
time,.016
iter,-80,,1                                 * use TSO
kbc,1
f,8,fy,-6.57e5
afwr
fini
/inp,27
fini
c*** Postprocess time-history results
/post26
disp,2,8,UY,Disp                            * tip dispacement
rfor,3,1,MZ,Mom                             * moment at fixed end
estr,4,8,8,Stat                             * gap status
estr,5,8,1,Forc                             * force in restraint
estr,6,8,7,Slid                             * amt of inelastic disp in restraint
prva,2,3,4,5,6
/graph,grid,1                               * place grid on graph plots
/graph,laby,DISP
plva,2,6
/graph,laby,MOM
plva,3
fini
c*** display deformed shapes
```

```
/post1
set,,,,,.003                                    * retrieve results by time
/dsca,,10                                       * set disp. scaling for all times
pldi,1
/noer                                           * overlay results
set,,,,,.0056
pldi
set,,,,,.008
pldi
set,,,,,.012
pldi
set,,,,,.016
pldi
c*** use /tlabel to label finished display
c***    format: /tlabel,,,label (done interactively with picking device)
fini
```

4.3 Large Displacement Elastic Dynamic Response of Clamped Beam

```
/PREP7
/TITLE, LARGE DISPLACEMENT ELASTIC DYNAMIC RESPONSE OF A CLAMPED BEAM
C*** From "Evaluation of Solution Schemes for Nonlinear Structures",
C***    D.P.Mondkar and G.H.Powell, Computers and Structures No. 9, p 223
KAN,4               * Nonlinear Transient Dynamic Analysis
KAY,5,2             * Zero initial velocity and acceleration
KAY,6,1             * Large Deflection
ET,1,82
MP,EX,1,30E6           * psi
MP,NUXY,1,0
MP,DENS,1,2.53623E-4   * lb*s**2/in**4
N,1
N,3,,.125              * in
FILL
N,4,1
N,5,1,.125
NGEN,6,5,1,5,1,2
NDEL,29,30,1
E,1,6,8,3,4,7,5,2
EGEN,5,5,1
D,1,ALL,,,3,1          * Clamp end of beam
SYMBC,,1,10            * Symmetry at center of beam
KBC,1                  * Step load
F,28,FY,-320
ITER,-1000,0,1         * Use time step optimization
TIME,.005
AFWRITE
FINISH
/INPUT,27
FINISH
/POST26
C*** Retrieve deflection and response frequency history
DISP,2,27,UY           * Central deflection
```

```
ESTR,3,-1,19,FREQ       * Response frequency
ESTR,4,-1,9,NEQ         * Nunber of equlibrium iterations
ESTR,5,-1,26,DTIM       * Integration time step
PROD,2,2,,,DEFL,,,-1    * Reverse sign on UY
PROD,5,5,,,DTIM,,,1E6   * Convert time step to microseconds
PROD,6,1,,,TIME,,,1E6   * Convert time to microseconds
PRVAR,6,2,3,4,5
XVAR,6
/GRAPH,LABX,TIME
/GRAPH,LABY,DEFL
PLVAR,2
/GRAPH,LABY,FREQ
PLVAR,3
/GRAPH,LABY,NEQ
YRANGE,0,5
PLVAR,4
/GRAPH,LABY,DTIM
YRANGE,0,200
PLVAR,5
FINI
```

4.4 Impulsively Loaded Simply Supported Plastic Beam

```
/PREP7
/TITLE, IMPULSIVELY LOADED SIMPLY SUPPORTED PLASTIC BEAM
C*** From "Implicit Finite Element Methods," M. Geradin, M. Hogge, and
C***   S. Idelsohn, COMPUTATIONAL METHODS FOR TRANSIENT ANALYSIS,
C***   T.B. Belytschko and T.J.R. Hughes, Ed., 1983, p. 462.
KAN,4                   * Nonlinear Transient Dynamic Analysis
KAY,5,2                 * Initial velocity and acceleration are zero
KAY,9,3                 * Newton-Raphson option key, zero allows default
GAMMA,,8                * Set number of equilibrium iterations
ET,1,82
MP,EX,1,3E7
MP,NUXY,1,0.3
MP,DENS,1,.733E-03
KNL,1
NL,1,13,10              * Classical Bilinear Kinematic Hardening
NL,1,25,50000           * Yield stress, tangent modulus = 0
N,1
N,3,,1
FILL
N,4,1.5
N,5,1.5,1
NGEN,6,5,1,5,1,3
NDEL,29,30,1
E,1,6,8,3,4,7,5,2
EGEN,5,5,1
D,1,UX,,,,,UY           * Pinned at end
SYMBC,,1,15             * Symmetry at beam center
ASYMBC,,2,0             * Asymmetry at centerline
PSF,,2,1,166.5          * Pressure along top surface of beam
```

```
ITER,-1000,0,1
TIME,.012
KBC,1
POST,1,1,4
AFWRIT
FINISH
/INPUT,27
FINISH
/POST26
C*** Retrieve deflection history
DISP,2,27,UY
PROD,2,2,,,,,,-1     * Reverse sign on UY
PROD,3,1,,,,,,1000   * Convert to microseconds
PRVA,2
/GRAPH,LABX,TIME
/GRAPH,LABY,DEFL
XVAR,3
PLVA,2
FINI
```

4.5 Slider – Crank Mechanisme Under Impact

```
/PREP7
/TITLE, SLIDER-CRANK MECHANISM UNDER IMPACT
/COM From Khulief, Y.A., and Shabana, A.A., "Dynamic Analysis of Constrained
/COM   System of Rigid and Flexible Bodies With Intermittent Motion",
/COM   Trans of the ASME, Vol 108, March 1986 p38-45.
KAN,4                       * Nonlinear Transient Dynamic
KAY,5,0                     * Zero initial acceleration and nonzero velocity
KAY,6,1                     * Large deflection
PI=(4*ATAN(1.0))
TOTT=((4*PI)/150)           * Total time of transient (2 revolutions)
ET,1,3
ET,2,21,,,4
ET,3,40,,,,,,2
BETAD,5.3E-6                          * Structural damping on stiffness
MP,EX,1,2E11                          * Pascal, connecting rod
MP,DENS,1,7.87E3                      * kg/m**3
MP,EX,2,2E11                          * Pascal, crank
MP,DENS,2,1E-5                        * Small density on massless crank
R,1,2.827433E-5,6.361725E-11,.006     * Circular section
R,2,0.033377851                       * Mass half of connecting rod
R,3,1E8,,0.166889256,.383             * Stiffness, Block D mass, gap
R,4,2.83E-3,6.36E-7,.06               * Circular section, rigid crank
N,1
N,2,.15
N,3,.15
N,4,.3
N,5,.45
N,6,1.0                               * Node 6 location is arbitrary
TYPE,1  $MAT,2  $REAL,4               * Crank
E,1,2
```

```
MAT,1  $REAL,1                          * Connecting rod
E,3,4
E,4,5
TYPE,2  $REAL,2                         * Slider B
E,5
TYPE,3  $REAL,3                         * Gap and Block D
E,5,6
CP,1,UX,2,3                             * Coupling for hinge joint
CP,2,UY,2,3
C*** First load step - set Block D (mass) in motion
TLS=(TOTT/2000)                         * Time of this load step (10 time steps)
TIME,TLS
ITER,10,,1
D,1,UX,,,,,UY                           * Crank pivot
D,5,UY,,,6,1                            * Sliders
D,1,ROTZ,(150*TLS)                      * Will lead to crank speed of 150 r/s
D,6,UX,-(15*TLS)                        * Will lead to slider velocity of 15 m/s
LWRITE
C*** Second load step - allow free motion of Block D (mass)
TIME,TOTT                               * Two revolutions of the crank
ITER,-19990,,1                          * TSO with minimum time step of 4.2E-6 seconds
D,1,ROTZ,(150*TOTT)
DDELE,6,UX
LWRITE
AFWRIT
FINISH
/INPUT,27
FINISH
/POST26
C*** Display velocity and force time histories
DISP,2,1,ROTZ
DISP,3,5,UX
DISP,4,6,UX
NFORCE,5,3,4,FX
NFORCE,6,3,4,FY
DERIV,7,2,1,,CRAN,K_W
DERIV,8,3,1,,TIP_,VELO
DERIV,9,4,1,,MASS,VELO
XVAR,2                          * X-axis is crank angle (radians)
/GRAPH,LABX,ROTA
/GRAPH,LABY,VELO
PLVAR,8,9                       * Display velocity of sliders
YRAN,-750,750                   * Limit force displays to ignore noise
/GRAPH,LABY,FX
PLVAR,5
/GRAPH,LABY,FY
PLVAR,6
C*** Display the transverse displacement of the midpoint of the crank
C***  Use vector algebra to determine the relative displacement with respect
C***  to the undeformed, but rotated, position (line AB)
RESET
NUMVAR,25
DISP,25,1,ROTZ
```

```
C*** Retrieve the displacements at the middle and end of the link
DISP,2,3,UX,Vx3
DISP,3,3,UY,Vy3
DISP,4,4,UX,Vx4
DISP,5,4,UY,Vy4
DISP,6,5,UX,Vx5
DISP,7,5,UY,Vy5
C*** Determine the relative displacement of the center of the link
C***    V = 0.5*(V3 + V5) - V4
ADD,8,2,6,,,,,0.5,0.5
ADD,9,3,7,,,,,0.5,0.5
ADD,10,8,4,,Vx,,,1,-1
ADD,11,9,5,,Vy,,,1,-1
PROD,15,10,10,,Vx,sqrd
PROD,16,11,11,,Vy,sqrd
ADD,17,15,16,,Vx2p,Vy2
SQRT,18,17,,,V
C*** Determine updated coordinates of nodes 3 and 5
C***    Xn53 = (X5+V5) - (X3+V3)
FILL,12,,,,.3                  * Original coordinate of node 5 relative to node 3
ADD,13,12,6,2,Xn,53x,,1,1,-1
ADD,14,7,3,,Xn,53y,,1,-1
C*** Get direction of transverse displacement from the cross product
C***    direction = V x Xn53
PROD,19,13,11,,XxVy
PROD,20,14,10,,XyVx
ADD,21,19,20,,Xn53,xV,,1,-1
ABS,22,21,,,mXn5,3xV
QUOT,23,21,22,,,sign              * Sign of transverse displacement, + or - 1.0
PROD,24,23,18,,,Beta,,-3.33333    * Make dimensionless by dividing by length (.3)
XVAR,25
/GRAPH,LABX,ROTA
/GRAPH,LABY,BETA
PLVA,24
FINISH
```

5. The Four – Node Plastic Shell Element STIF 43

5.1 Patch Test

```
/VERIFY,STIF43-1
/PREP7
/TITLE, STIF43-1 , PATCH TEST - STIF43
/COM,  REF. R.H. MACNEAL ET. AL. , 'A PROPOSED STANDARD SET OF PROBLEMS
/COM,  TO TEST FE ACCURACY', FE IN ANALYSIS AND DESIGN I, APRIL, 1985
ET,1,43
C***   ELEMENT THICKNESS AND MATERIAL PROPERTIES
R,1,.001
EX,,1E6   $NUXY,,.25
C***  NODES AND ELEMENTS
N,1   $N,2,.24   $N,3,.04,.02   $N,4,.18,.03   $N,5,.08,.08
N,6,.16,.08   $N,7,,.12   $N,8,.24,.12
REAL,1
E,1,2,4,3
E,6,5,3,4
E,8,7,5,6
E,1,3,5,7
E,2,8,6,4
ITER,1,0   $KRF,2
/TITLE, PATCH TEST - STIF43 - MEMBRANE (SX=SY=1333, SXY=400)
D,1,ALL
D,2,UX,.24E-3   $D,2,UY,.12E-3
D,7,UX,.06E-3   $D,7,UY,.12E-3
D,8,UX,.30E-3   $D,8,UY,.24E-3
LWRITE
/TITLE, PATCH TEST - STIF43 - BENDING (SX=SY=.667, SXY=.200)
DDEL,1,ALL,8
D,1,ALL  $D,2,UY
D,2,UZ,.0288E-3   $D,2,ROTX,.12E-3   $D,2,ROTY,-.24E-3
D,7,UZ,.0072E-3   $D,7,ROTX,.12E-3   $D,7,ROTY,-.06E-3
D,8,UZ,.0504E-3   $D,8,ROTX,.24E-3   $D,8,ROTY,-.30E-3
LWRITE
AFWRITE   $FINISH
/INP,27
FINISH
/POST1
SET,1,1
PRESTR  $PRSTRS
TOP     $GLOBAL,1
PRNSTR
SET,2,1
PRESTR  $PRSTRS
TOP     $GLOBAL,1
PRNSTR
FINISH
```

5.2 Simply Supported .....Element Performance for Thick to Moderately Thi

```
/VERIFY,STIF43-2
C**********
C***  DEFINE PARAMETERS
C**********
LTHC = 1
THIC = .0001
ELMD = 10E6
POSN = 0.3
Q    = 1.0
L    = 10.0
C**********
C***  BEGIN PREP7 PREPROCESSING
C**********
/PREP7
/TITLE,STIF43-2, SIMPLY SUPPORTED SQUARE PLATE UNDER UNIFORM PRESSURE LOAD
/COM,  UNIFORM MESH
/COM,  REF.  TIMOSHENKO, PP 120
C*************************************************************************************
C***                                                                               ***
C***  MAXIMUM DISPLACEMENT = 0.00406*Q*(L**4)/D                                    ***
C***                                                                               ***
C***                       D = (EX*T**3)/12(1-NUXY**2)                             ***
C***                                                                               ***
C*************************************************************************************
ET,1,43                      * STIF43 - PLASTIC QUADRILATERIAL SHELL ELEMENT
C**********
C***  MATERIAL PROPERTIES
C**********
EX,1,ELMD
NUXY,1,POSN
C**********
C***  DEFINE ELEMENT THICKNESS
C**********
R,1,THIC
C**********
C***  CALCULATE NORMALIZED MAXIMUM DEFLECTION
C**********
DTM1 = (ELMD*(THIC**3))
DTM2 = (12*(1 - (POSN**2)))
DTEM = DTM1/DTM2
DTH  = ((0.00406*Q)*(L**4))/DTEM
C**********
C***  CREATE NODES AND ELEMENTS
C**********
N,1
N,3,5
FILL
NGEN,3,10,1,3,1,,2.5
E,1,2,12,11
EGEN,2,1,-1
EGEN,2,10,-2
```

```
C***********
C***  SET SYMETRICAL BOUNDRY CONDITIONS
C***********
SYMBC,,1
SYMBC,,2
NSEL,X,5
D,ALL,UZ
D,ALL,ROTX
NALL
NSEL,Y,5
D,ALL,UZ
D,ALL,ROTY
NALL
ITER,1,0,1
C***********
C***  SPECIFY PRESSURE ON FACE 2 OF ELEMENTS 1 THROUGH 16
C***********
EP,1,2,-1.0,,4,1
C***********
C*** WRITE AND INPUT FILE27 FOR EXECUTION
C***********
AFWRITE
FINISH
/INPUT,27
FINISH
C***********
C***  BEGIN POST1 POSTPROCESSING
C***********
/POST1
SET,1,1
*GET,DMAX,UZ,1
RAT = DMAX/DTH
*STAT
FINISH
C***********
C***  BEGIN DESIGN OPTIMIZATION
C***********
/OPT
THIC = THIC*10
LTHC = LTHC+1
OPCOPY
:LOP
RUN
*IF,LTHC,LE,4,:LOP
PRVAR,RAT
FINISH
```

5.3 Element Performance Fixed with Uniform Load

```
/VERIFY,STIF43-2
C***********
C***  DEFINE PARAMETERS
C***********
LTHC = 1
THIC = .0001
ELMD = 10E6
POSN = 0.3
Q    = 1.0
L    = 10.0
C***********
C***  BEGIN PREP7 PREPROCESSING
C***********
/PREP7
/NOPR
/TITLE,STIF43-2 - FIXED FIXED SQUARE PLATE UNDER UNIFORM PRESSURE LOAD
/COM,  UNIFORM MESH
/COM,  REF.  TIMOSHENKO, PP 202
C*****************************************************************************************
C***                                                                                   ***
C***  MAXIMUM DISPLACEMENT = 0.00126*Q*(L**4)/D                                        ***
C***                                                                                   ***
C***                        D = (EX*T**3)/12(1-NUXY**2)                                ***
C***                                                                                   ***
C*****************************************************************************************
ET,1,43
C***********
C***  MATERIAL PROPERTIES
C***********
EX,1,ELMD
NUXY,1,POSN
C***********
C***  DEFINE ELEMENT THICKNESS
C***********
R,1,THIC
C***********
C***  CALCULATE NORMALIZED MAXIMUM DEFLECTION
C***********
DTM1 = (ELMD*(THIC**3))
DTM2 = (12*(1 - (POSN**2)))
DTEM = DTM1/DTM2
DTH  = ((0.00126*Q)*(L**4))/DTEM
C***********
C***  CREATE NODES AND ELEMENTS
C***********
N,1
N,3,5
FILL
NGEN,3,10,1,3,1,,2.5
E,1,2,12,11
EGEN,2,1,-1
```

```
EGEN,2,10,-2
C**********
C***  SET SYMETRICAL BOUNDRY CONDITIONS
C**********
SYMBC,,1
SYMBC,,2
NSEL,X,5
D,ALL,ALL
NALL
NSEL,Y,5
D,ALL,ALL
NALL
ITER,1,0,1
C**********
C***   SPECIFY PRESSURE ON FACE 2 OF ELEMENTS 1 THROUGH 16
C**********
EP,1,2,-1.0,,4,1
C**********
C*** WRITE AND INPUT FILE27 FOR EXECUTION
C**********
AFWRITE
FINISH
/INPUT,27
FINISH
C**********
C***  BEGIN POST1 POSTPROCESSING
C**********
/POST1
SET,1,1
*GET,DMAX,UZ,1
RAT = DMAX/DTH
*STAT
FINISH
C**********
C***  BEGIN DESIGN OPTIMIZATION
C**********
/OPT
THIC = THIC*10
LTHC = LTHC+1
OPCOPY
:LOP
RUN
*IF,LTHC,LE,4,:LOP
PRVAR,RAT
FINISH
```

5.4 Element Performance : Simply Supported Distorted Mesh

```
/VERIFY,STIF43-2
C***********
C***  DEFINE PARAMETERS
C***********
LTHC = 1
THIC = .0001
ELMD = 10E6
POSN = 0.3
Q    = 1.0
L    = 10.0
C***********
C***  BEGIN PREP7 PREPROCESSING
C***********
/PREP7
/TITLE,STIF43-2, SIMPLY SUPPORTED SQUARE PLATE UNDER UNIFORM PRESSURE LOAD
/COM,  DISTORTED MESH
/COM,  REF.  TIMOSHENKO, PP 120
C*************************************************************************************
C***                                                                                ***
C***  MAXIMUM DISPLACEMENT = 0.00406*Q*(L**4)/D                                     ***
C***                                                                                ***
C***                        D = (EX*T**3)/12(1-NUXY**2)                             ***
C***                                                                                ***
C*************************************************************************************
ET,1,43                 * STIF43 - PLASTIC QUADRILATERIAL SHELL ELEMENT
C***********
C***  MATERIAL PROPERTIES
C***********
EX,1,ELMD
NUXY,1,POSN
C***********
C***  DEFINE ELEMENT THICKNESS
C***********
R,1,THIC
C***********
C***  CALCULATE NORMALIZED MAXIMUM DEFLECTION
C***********
DTM1 = (ELMD*(THIC**3))
DTM2 = (12*(1 - (POSN**2)))
DTEM = DTM1/DTM2
DTH  = ((0.00406*Q)*(L**4))/DTEM
C***********
C***  CREATE NODES AND ELEMENTS
C***********
N,1
N,5,5
FILL
NGEN,5,10,1,5,1,,1.25
N,12,1.375,1.375
N,13,2.75,1.5
N,14,3.875,1.375
```

```
N,22,1.5,2.75
N,23,3,3
N,24,4,2.75
N,32,1.375,3.875
N,33,2.75,4
N,34,3.875,3.875
E,1,2,12,11
EGEN,4,1,-1
EGEN,4,10,-4
C**********
C***  SET SYMETRICAL BOUNDRY CONDITIONS
C**********
SYMBC,,1
SYMBC,,2
D,ALL,ROTZ
NSEL,X,5
D,ALL,UZ,
D,ALL,ROTX
NALL
NSEL,Y,5
D,ALL,UZ
D,ALL,ROTY
NALL
ITER,1,0,1
C**********
C***  SPECIFY PRESSURE ON FACE 2 OF ELEMENTS 1 THROUGH 16
C**********
EP,1,2,-1.0,,16,1       * SPECIFY PRESSURE ON FACE 2 OF ELEMENTS 1 THROUGH 16
C**********
C*** WRITE AND INPUT FILE27 FOR EXECUTION
C**********
AFWRITE
FINISH
/INPUT,27
FINISH
C**********
C***  BEGIN POST1 POSTPROCESSING
C**********
/POST1
SET,1,1
*GET,DMAX,UZ,1
RAT = DMAX/DTH
*STAT
FINISH
C**********
C***  BEGIN DESIGN OPTIMIZATION
C**********
/OPT
THIC = THIC*10
LTHC = LTHC+1
OPCOPY
:LOP
RUN
```

```
*IF,LTHC,LE,4,:LOP
PRVAR,RAT
FINISH
```

5.5 Element Performance : Fixed , Distorted Mesh

```
/VERIFY,STIF43-2
C**********
C***  DEFINE PARAMATERS
C**********
LTHC = 1
THIC = .0001
ELMD = 10E6
POSN = 0.3
Q    = 1.0
L    = 10.0
C**********
C***  BEGIN PREP7 PREPROCESSING
C**********
/PREP7
/TITLE,STIF43-2, FIXED FIXED SQUARE PLATE UNDER UNIFORM PRESSURE LOAD
/COM,  DISTORTED MESH
/COM,  REF.  TIMOSHENKO, PP 202
C*************************************************************************************
C***                                                                              ***
C***  MAXIMUM DISPLACEMENT = 0.00126*Q*(L**4)/D                                   ***
C***                                                                              ***
C***                D = (T**3)/12(1-NUXY**2)                                      ***
C***                                                                              ***
C*************************************************************************************
ET,1,43             * STIF43 - PLASTIC QUADRILATERAL SHELL ELEMENT
C**********
C**********
EX,1,ELMD
NUXY,1,POSN
C**********
C***  DEFINE ELEMENT THICKNESS
C**********
R,1,THIC
C**********
C***  CALCULATE NORMALIZED MAXIMUM DEFLECTION
C**********
DTM1 = (ELMD*(THIC**3))
DTM2 = (12*(1 - (POSN**2)))
DTEM = DTM1/DTM2
DTH  = ((0.00126*Q)*(L**4))/DTEM
C**********
C***  CREATE NODES AND ELEMENTS
C**********
N,1
N,5,5
FILL
```

```
NGEN,5,10,1,5,1,,1.25
N,12,1.375,1.375
N,13,2.75,1.5
N,14,3.875,1.375
N,22,1.5,2.75
N,23,3,3
N,24,4,2.75
N,32,1.375,3.875
N,33,2.75,4
N,34,3.875,3.875
E,1,2,12,11
EGEN,4,1,-1
EGEN,4,10,-4
C***********
C***  SET SYMETRICAL BOUNDRY CONDITIONS
C***********
SYMBC,,1
SYMBC,,2
NSEL,X,5
D,ALL,ALL,
NALL
NSEL,Y,5
D,ALL,ALL
NALL
ITER,1,0,1
C***********
C***  SPECIFY PRESSURE ON FACE 2 OF ELEMENTS 1 THROUGH 16
C***********
EP,1,2,-1.0,,16,1
C***********
C***   WRITE AND INPUT FILE27 FOR EXECUTION
C***********
AFWRITE
FINISH
/INPUT,27
FINISH
C***********
C***  BEGIN POST1 POSTPROCESSOR
C***********
/POST1
SET,1,1
*GET,DMAX,UZ,1
RAT = DMAX/DTH
*STAT
FINISH
C***********
C***  BEGIN DESIGN OPTIMIZATION
C***********
/OPT
THIC = THIC*10
LTHC = LTHC+1
OPCOPY
:LOP
```

```
RUN
*IF,LTHC,LE,4,:LOP
PRVAR,RAT
FINISH
```

5.6 Thick and Thin Cantilever Beam

```
/VERIFY,STIF43-3
C***  DEFINE PARAMETER
THIK = 0.1
/PREP7
/OUT,26
/TITLE, STIF43-3,  TEST STIF43, STIF45 AND STIF63
ET,1,43
ET,2,63
ET,3,45
R,1,THIK
C****   COMPUTE ACTUAL VALUES USING PARAMETRIC MATH
B=3
LEN=20
EX=30E6
DZ=.01
I = (B*(THIK**3))/12
AS = (B*THIK)*(5/6)
C**** REACTION FORCE (P) = ((DZ*EX)/LEN)/(((LEN**2)/(3*I)) + (2/AS))
/COM,  REF. R.J. ROARK ET. AL., 'FORMULAS FOR STRESS AND STRAIN',
/COM,  MCGRAW-HILL, 5TH ED. PP185, 1975
RFOR = ((DZ*EX)/LEN)/(((LEN**2)/(3*I)) + (2/AS))
EX,,30E6
NUXY,,0
N,1
N,2,,3
NGEN,5,2,1,2,1,5
NGEN,4,10,1,10,1,,,THIK
E,1,3,4,2
EGEN,4,2,-1
TYPE,2
E,11,13,14,12
EGEN,4,2,-1
TYPE,3
E,21,23,24,22,31,33,34,32
EGEN,4,2,-1
ITER,1
PRRF,1
NSEL,X
D,ALL,ALL
NALL
NSEL,X,20
D,ALL,ROTX
D,ALL,UZ,.01
NALL
AFWRITE
```

```
FINISH
/INPUT,27
FINISH
/OUT,6
/POST1
SET,1,1
PRRFOR
*STAT
FINISH
C***  BEGIN OPTIMIZATION TO CHANGE ELEMENT THICKNESSES
/OPT
OPCOPY
THIK = 0.3
RUN
THIK = 1.0
RUN
THIK = 3.0
RUN
THIK = 10.0
RUN
FINISH
```

5.7 Linear Analysis of a Cylindrical (Scordelis – LO) Shell

```
/VERIFY,STIF43-4
C***  DEFINE PARAMETERS
NTIM = 1
MDEN = 4
/PREP7
/OUT,26
/TITLE, STIF43-4, SCORDELIS-LO ROOF WITH QUADRILATERAL ELEMENTS
/COM,  REF. E.N.DVORKIN, PH.D THESIS, MIT, 1984
ET,1,43
EX,1,0.432E+9
NUXY,1,0.0
R,1,0.25
DENS,1,0.36
*SET,NZ,MDEN+1
*SET,ANG,(40./MDEN)
CSYS,1
N,1,25,50
N,NZ,25,50,25
FILL
NGEN,NZ,NZ,1,NZ,1,,ANG
*SET,A1,1
*SET,A2,2
*SET,A3,3
*SET,A3,NZ+1
*SET,A4,A3+1
E,A1,A2,A4,A3
EGEN,MDEN,1,-1
EGEN,MDEN,NZ,-MDEN
```

```
CSYS,0
SYMBC,0,1,0
SYMBC,0,3,0
NSEL,Z,25
D,ALL,UX,0,,,,UY,ROTZ
NALL
ACEL,0,1000,0
ITER,1,0,1
WSORT,Z
AFWRITE
FINISH
/INPUT,27
FINISH
/OUT,6
/POST1
SET,1,1
NSEL,NODE,1
PRDI
FINISH
C***  BEGIN OPTIMIZATION TO REFINE MESHES
/OPT
MDEN = MDEN + 2
NTIM = NTIM + 1
OPCOPY
:LOP
RUN
*IF,NTIM,LE,5,:LOP
FINISH
```

5.8 Linear Analysis of a Pinched Cylinder

```
/VERIFY,STIF43-5
C***  DEFINE PARAMETERS
NTIM = 1
MDEN = 4
/PREP7
/OUT,26
/TITLE,STIF43-5,  PINCHED CYLINDER
/COM,  REF. G.M. LINDBERG ET. AL., 'NEW DEVELOPMENTS IN FEA OF SHELLS',
/COM,  QUAT. BULL. OF DIV. OF ENG. MECH. AND AERO., NRC CANADA, VOL.4 1969
ET,1,43   $R,1,3
EX,,3E6   $NUXY,,.3
CSYS,1
K,1,300,0   $K,2,300,90   $KGEN,2,1,2,1,,,-300
A,1,2,4,3
ELSI,,MDEN                              * MDEN X MDEN
AMESH,1
WSORT,Z
ITER,1
SYMBC,,1   $SYMBC,,2   $SYMBC,,3,-300
CSYS,1
NSEL,Z,0   $D,ALL,UX,,,,,UY,ROTZ   $NALL          *FIXED ALONG ARCHED EDGE
```

```
NSEL,Z,-300   $NRSEL,Y,90   $F,ALL,FY,-0.25   $NALL
SFWRITE   $FINISH
/GOPR
/SOLVE
FINISH
/OUT,6
/POST1
SET,1,1
CSYS,1
NSEL,Z,-300   $NRSEL,Y,90
CSYS,0
PRDI
NSORT,UY
*GET,D,MAX
R=D/(0.18248E-4)
*STAT
FINISH
C***  BEGIN OPTIMIZATION FOR MESH REFINEMENT
/OPT
MDEN = MDEN*2
NTIM = NTIM + 1
OPCOPY
:LOP
RUN
*IF,NTIM,LE,3,:LOP
FINISH
```

5.9 Free Vibration of Simply Supported Rectangular Plate

```
/VERIFY,STIF43-6
/PREP7
/TITLE, STIF43-6,  NATURAL FREQUENCIES OF S.S. RECTANGULAR PLATE
/COM,  REF. R.D. BLEVINS, FORMULAS FOR NATURAL FREQUENCY AND MODE SHAPES.
/COM,  VAN NOSTRAND REINHOLD CO., 1979
C*** CHECKS MASS MATRIX - CONSISTENT AND REDUCED - RECTANGLES AND TRIANGLES
C***   MODE #1=134.7, #2=466.0
KAN,2   $ET,1,43   $R,1,.1
ET,2,43
MP,EX,,10E6   $MP,NUXY,,.3   $MP,DENS,,2.6E-4
K,1   $K,2,7.5   $K,3,7.5,5   $K,4,,5
A,1,2,3,4
AGEN,2,1
C*** GENERATE 6X6 RECTANGULAR ELEMENTS
AATT,1,1,1  $ELSIZE,,6  $AMESH,1
C*** GENERATE TRIANGULAR ELEMENTS
AATT,1,1,2   $ELSIZE,,6,1   $AMESH,2
NSEL,EXT
NINV
M,ALL,UZ
TOTAL,60
ITER,1,,1   $KAY,2,4            * 1ST 4 MODES EXPANDED
NSEL,X,7.5   $NASEL,Y,5   $D,ALL,UZ   $NALL
```

```
SYMBC,,1   $SYMBC,,2
WAVES
SFWRITE   $FINISH
/SOLVE
FINI
/POST1
SET,1,1     * FIRST MODE FOR RECTANGULAR MESH
ESEL,TYPE,1
NELEM
PRDISP
SET,1,2     * FIRST MODE FOR TRIANGULAR MESH
ESEL,TYPE,2
NELEM
PRDISP
SET,1,3
NINV
PRDISP
SET,1,4
FINI
```

5.10 Vibration of Prestressed Simply Supported Rectangular Plate

```
/VERIFY,STIF43-7
/PREP7
/TITLE, STIF43-7, NORMAL VIBRATION OF A STRETCHED RECTANGULAR MEMBRANE
/COM CHECKS MASS MATRIX AND STRESS STIFFENING
/COM    1ST MODE = 222HZ, 2ND MODE = 413HZ
/COM,  REF. R.D. BLEVINS, FORMULAS FOR NATURAL FREQUENCY AND MODE SHAPES.
/COM,  VAN NOSTRAND REINHOLD CO., 1979
ET,1,43
R,1,.0001
MP,EX,,30E6   $MP,NUXY,,.3   $MP,DENS,,.00073
N,1   $N,7,7.5   $FILL   $NGEN,7,10,1,7,1,,0.83333333
E,1,2,12,11   $EGEN,6,1,-1   $EGEN,6,10,-6
ITER,1,0,1
KRF,2
SYMBC,,1   $SYMBC,,2
D,7,UZ,,,67,10   $D,61,UZ,,,66
C*** APPLY EDGE PRESSURES TO INTRODUCE UNIFORM STRESS STATE
PSF,0,1,7.5,-10000
PSF,0,2,5,-10000
AFWR   $FINI
/INP,27
FINI
C*** FULL SUBSPACE, PRINT 1ST 2 MODES, INCLUDE STRESS STIFFENING
/FREQ,,10,2,,1,1
ITER,1,1
END
FINI
```

5.11 Vibration Behaveour of Twisted Cantilever Plates

```
/VERIFY,STIF43-8
*CREATE,RUN
/PREP7
/TITLE,STIF43-8,  VIBRATION OF TWISTED CANTILEVER PLATES
/COM,  REF. J.C. MACBAIN, J.AIRCRAFT, VOL. 12, 1975
KAN,2
KAY,2,8
ET,1,43
EX,1,1E7
NUXY,1,0.3
R,1,0.09
DENS,1,2.587E-4
N,1,-1.5
N,9,1.5
FILL
CSYS,1
NGEN,17,20,1,9,1,,ARG1,0.4375
E,1,21,22,2
EGEN,8,1,-1
EGEN,16,20,-8
CSYS,0
NROT,ALL
ITER,1,1,1
NSEL,Z
D,ALL,ALL
NALL
PRDIS,999,1,400              * SUPRESS ALL DISPLACEMENT PRINT OUT
TOTAL,40
AFWRITE
FINISH
*END
*USE,RUN,0
/INPUT,27
FINISH
*USE,RUN,1.0625
/INPUT,27
FINISH
*USE,RUN,1.875
/INPUT,27
FINISH
/SHOW,PR8,F33,1
/POST1
/VIEW,1,,1
/VUP,,Z
/EDGE,,1
/DSCALE,,OFF
/NUMBER,-1
SET,1,1
PLNSTR,UY
SET,1,2
PLNSTR,UY
```

```
SET,1,3
PLNSTR,UY
SET,1,4
PLNSTR,UY
SET,1,5
PLNSTR,UY
SET,1,6
PLNSTR,UY
FINISH
*USE,RUN,2.375
/INPUT,27
FINISH
```

5.12 Twisted Plate Vibration Under Centrifugal Loading

```
/VERIFY,STIF43-9
*CREATE,RUN
/PREP7
/TITLE,STIF43-9,  TWISTED PLATE VIBRATION UNDER CENTRIFUGAL LOAD
/COM,  REF. J.C. MACBAIN, J. AIRCRAFT, VOL. 12, 1975
ET,1,43
EX,1,1E7
NUXY,1,0.3
R,1,0.09
DENS,1,2.587E-4
N,1,-1.5,,7
N,9,1.5,,7
FILL
CSYS,1
NGEN,17,20,1,9,1,,-1.875,0.4375
E,1,21,22,2
EGEN,8,1,-1
EGEN,16,20,-8
CSYS,0
NROT,ALL
ITER,1,0,1
NSEL,Z,7
D,ALL,ALL
NALL
OMEGA,ARG1
AFWRITE
FINISH
*END
*USE,RUN,0
/INPUT,27
FINISH
/FREQ,40,,20,,1
ITER,1,1,1
OMEGA,ARG1,,,1
PRDIS,999,1,400              * SUPRESS DISPLACEMENT PRINT OUT
END
FINISH
```

```
*USE,RUN,418.88
/INPUT,27
FINISH
/FREQ,40,,20,,1
ITER,1,1,1
OMEGA,ARG1,,,1
PRDIS,999,1,400              * SUPRESS DISPLACEMENT PRINT OUT
END
FINISH
*USE,RUN,837.76
/INPUT,27
FINISH
/FREQ,40,,20,,1
ITER,1,1,1
OMEGA,ARG1,,,1
PRDIS,999,1,400              * SUPRESS DISPLACEMENT PRINT OUT
END
FINISH
*USE,RUN,1256.64
/INPUT,27
FINISH
/FREQ,40,,20,,1
ITER,1,1,1
OMEGA,ARG1,,,1
PRDIS,999,1,400              * SUPRESS DISPLACEMENT PRINT OUT
END
FINISH
/POST1
/SHOW,PR9,F33,1
/VIEW,1,,1
/VUP,,Z
/EDGE,,1
/DSCALE,,OFF
/NUMBER,-1
SET,1,1
PLNSTR,UY
SET,1,2
PLNSTR,UY
SET,1,3
PLNSTR,UY
SET,1,4
PLNSTR,UY
SET,1,5
PLNSTR,UY
SET,1,6
PLNSTR,UY
FINISH
*USE,RUN,1675.52
/INPUT,27
FINISH
/FREQ,40,,20,,1
ITER,1,1,1
OMEGA,ARG1,,,1
```

```
PRDIS,999,1,400              * SUPRESS DISPLACEMENT PRINT OUT
END
FINISH
```

5.13 Buckling of a Hinged – Hinged Semi – Circular Arc

```
/VERIFY,STIF43-10
*CREATE,RUN
/PREP7
/TITLE, STIF43-10, BUCKLING OF A HINGED-HINGED SEMI-CIRCULAR ARCH
/COM,  REF. J.L. BATOZ, 'CURVED FE AND SHELL THEORIES WITH REFERENCE TO
/COM,  BUCKLING OF CIRCULAR ARCH', INT. J. NUM .METH. ENG., VOL. 14
/COM,  NO.5, PP 774-779, 1979
ET,1,43
R,1,1
EX,1,5.5056E5
NUXY,1,0
CSYS,1
N,1,120
N,2,120,,10
NGEN,ARG1,2,1,2,1,,(180/ARG1)
C1=ARG1*2
C1=C1+1
C2=C1+1
N,C1,120,179.999
N,C2,120,179.999,1
E,1,2,4,3
EGEN,ARG1,2,-1
NRSEL,Y,0
NASEL,Y,180
D,ALL,UX,,,,,UY,ROTX,UZ
NALL
EP,ALL,1,1
ITER,1,0,1
AFWRITE
FINISH
*END
*USE,RUN,20
/INPUT,27
FINISH
/BUCKLE,,8,4,,,1
ITER,1,1,1
END
FINISH
*USE,RUN,40
/INPUT,27
FINISH
/BUCKLE,,8,4,,,1
ITER,1,1,1
END
FINISH
/SHOW,PR10,F33,1
```

```
/PREP7
RESUME
/VIEW,1,1,2,3
/PBC,TDIS,1
/PBC,RDIS,1
EPLOT
FINISH
/POST1
/VIEW,1,1,2,3
SET,1,1
PLDI,1
FINISH
```

5.14 Buckling of a Closed Ring

```
/VERIFY,STIF43-11
*CREATE,RUN
/PREP7
/TITLE, STIF43-11,  BUCKLING OF A CLOSED CIRCULAR RING
/COM,  REF. H.A. MANG AND R.H. GALLAGHER, 'FEA OF THIN SHELLS OF GENERAL
/COM,  FORM FOR DISPLACEMENT-DEPENDENT LOADS', AMD-VOL.48, PP.65-82, 1981
ET,1,43
EX,1,5.5056E5
NUXY,1,0
R,1,1
CSYS,1
N,1,120
N,ARG1+1,120,90
FILL
NGEN,2,100,1,ARG1+1,1,,,10
NROT,ALL
E,1,2,102,101
EGEN,ARG1,1,-1
SYMBC,1,2
SYMBC,1,2,90
D,ALL,UZ
EP,1,2,1,,ARG1,1
ITER,1,0,1
AFWRITE
FINISH
*END
*USE,RUN,10
/INPUT,27
FINISH
/BUCKLE,,12,6,,,1
ITER,1,1,1
END
FINISH
*USE,RUN,20
/INPUT,27
FINISH
/BUCKLE,,12,6,,,1
```

```
ITER,1,1,1
END
FINISH
/SHOW,PR11,F33,1
/POST1
/VIEW,1,1,2,3
SET,1,1
PLDI,1
FINISH
```

5.15 Buckling of Hinged – Hinged Circular Cylindrical Shell

```
/VERIFY,STIF43-12
*CREATE,RUN
/PREP7
/TITLE, STIF43-12,  BUCKLING OF HINGED-HINGED CICULAR CYLINDRICAL SHELL
/COM,  REF. D.O. BRUSH AND B.O. ALMROTH, 'BUCKLING OF BARS PLATES AND
/COM,  SHELLS', MCGRAW-HILL, PP.165, 1975
ET,1,43
EX,1,2.4E7
NUXY,1,0.16
R,1,1.0
CSYS,1
K,1,100
K,2,100,45
K,3,100,45,314.16
K,4,100,  ,314.16
A,1,2,3,4
ELSI,,ARG1
AMESH,1
NROT,ALL
NSEL,Z,314.16
D,ALL,UX
NALL
SYMBC,1,2,0
SYMBC,1,2,45
SYMBC,1,3
EP,1,2,10,,ARG1*ARG1,1
ITER,1,0,1
AFWRITE
FINISH
*END
*USE,RUN,3
/INPUT,27
FINISH
/BUCKLE,,12,6,0,,1
ITER,1,1,1
END
FINISH
*USE,RUN,6
/INPUT,27
FINISH
```

```
/BUCKLE,,12,6,0,,1
ITER,1,1,1
END
FINISH
*USE,RUN,12
/INPUT,27
FINISH
/BUCKLE,,12,6,0,,1
ITER,1,1,1
END
FINISH
/SHOW,PR12,F33,1
/POST1
/VIEW,1,1,2,6
SET,1,1
PLDI,1
FINISH
```

5.16 Large Deflection Analysis of a Clamped Square Plate

```
/VERIFY,STIF43-13
/PREP7
/TITLE, STIF43-13, LARGE DEFLECTION OF A CLAMPED SQUARE PLATE UNDER
/COM,  UNIFORM TRANSVERSE LOAD
/COM,  REF. O.C. ZIENKIEWICZ, 'THE FINISHTE ELEMENT METHOD',
/COM,  MCGRAW-HILL, PP.508, 1977
ET,1,43
EX,1,10E6
NUXY,1,.3
R,1,.1
KAY,6,1
K,1
K,2,10
K,3,10,10
K,4,,10
A,1,2,3,4
ELSI,,4,0
AMESH,1
WSORT,X
SYMBC,,1
SYMBC,,2
NRSEL,X,10
D,ALL,ALL
NALL
NRSEL,Y,10
D,ALL,ALL
NALL
ITER,-40
EP,1,1,0,,100
LWRITE
EP,1,1,.457875,,100
LWRITE
```

```
EP,1,1,.915750,,100
LWRITE
EP,1,1,1.373626,,100
LWRITE
EP,1,1,1.831501,,100
LWRITE
EP,1,1,2.289377,,100
LWRITE
ITER,-40,40
EP,1,1,2.747252,,100
LWRITE
AFWRITE
FINISH
/INPUT,27
FINISH
/SHOW,PR13,F33,1
/POST26
TVAR,1
DISP,2,1,UZ,WC
ADD,3,2,,,DISP,,,10
ESTR,4,1,223,PRES
ADD,5,4,,,PRES,,,109.2
PRVAR,2,3,4,5
/GRAPH,GRID,1
/GRAPH,LABX,DISP
/GRAPH,LABY,LOAD
XVAR,3
PLVAR,5
FINISH
```

5.17 Simply Supported Square Plate Uniform Impulse Load

```
/SHOW,FILE33,DAT
/VERIFY,STIF43-14
/PREP7
/TITLE, STIF43-14, SIMPLY SUPPORTED RECTANGULAR PLATE UNDER UNFORM IMPULSE
/COM,  LOAD,  FROM "EXPLICIT ALGORITHMS FOR NONLINEAR DYNAMICS OF SHELLS",
/COM,  T.B.BELYTSCHKO AND C.S.TSAY, IN "NONLINEAR FINITE ELEMENT ANALYSIS OF
/COM,  PLATES AND SHELLS", T.R.J.HUGHES ET AL ED., AMD-48, ASME, P209-231, 1981
KAN,4
KAY,5,2                                 * START AT REST
ET,1,43                                 * QUADRILATERAL SHELL ELEMENT
R,1,0.5
C*** DEFINE MATERIAL PROPERTIES
EX,1,1E7
DENS,1,2.588E-4
NUXY,1,0.3
KNL,1
NL,1,13,10                              * C13=10, BILINEAR KINEMATIC HARDENING
NL,1,25,30000                           * YIELD STRESS=30000, TANGENT MODULUS=0
C*** DEFINE GEOMETRY
K,1
```

```
K,2,5
K,3,5,5
K,4,,5
A,1,2,3,4
ELSIZE,,4,2                          * 4 ELEMENTS ON A SIDE, QUAD ELEMENTS
AMESH,1
WSORT,X
C***                                 * USE DEFAULT CONVERGENCE CRITERIA
ITER,30,0,1
POSTR,1,1,5                          * PUT ELEMENT STRAINS ON POST DATA FILE
TIME,1.2E-3
KBC,1
NSEL,X,5
D,ALL,UZ,,,,,ROTX                    * SIMPLY SUPPORTED ON EDGES
NSEL,Y,5
D,ALL,UZ,,,,,ROTY
NALL
SYMBC,0,1    $SYMBC,0,2
PSF,0,3,0,300                        * PRESSURE=300 (STEPPED)
AFWRITE
FINISH
/INPUT,27
FINISH
/SHOW,PR14,F33,1
/POST26
C*** PLOT DEFLECTION TIME HISTORY
DISP,2,1,UZ
ESTR,3,-1,20,PLCR                    * GET PLASTICITY CONVERGENCE VALUE
PRVAR,2,3
/GRAPH,GRID,1
/GRAPH,LABX,TIME
/GRAPH,LABY,DISP
PLVAR,2                              * PLOT DEFLECTION
/GRAPH,LABY,PLCR
PLVAR,3                              * PLOT PLASTICITY CONVERGENCE VALUE
FINISH
```

5.18 Nonlinear Impulsively Loaded Clamped Plate

```
/VERIFY,STIF43-15
/PREP7
/TITLE, STIF43-15, NONLINEAR IMPULSIVELY LOADED CLAMPED PLATE
/COM, FROM "EXPLICIT ALGORITHMS FOR NONLINEAR DYNAMICS OF SHELLS",
/COM,    T.B.BELYTSCHKO AND C.S.TSAY, IN "NONLINEAR FINISHTE ELEMENT ANALYSIS OF
/COM,    PLATES AND SHELLS", T.R.J.HUGHES ET AL ED., AMD-48, ASME, P209-231, 1981
C*** DEFINE ANALYSIS OPTIONS
KAN,4
KAY,5,0                              * DEFINE INITIAL VELOCITY
KAY,6,1                              * LARGE DEFLECTION
GAMMA,,8                             * UP THE NO. OF EQUILIBRIUM ITERATIONS
ET,1,43
R,1,.125
```

```
C*** DEFINE MATERIAL PROPERTIES
EX,,10.4E6
NUXY,,.3
DENS,,2.61E-4
KNL,1
NL,,13,10                              * C13=10, BILINEAR KINEMATIC HARDENING
NL,,25,41400                           * YIELD=41400, TANGENT MODULUS=0
C*** DEFINE GEOMETRY - MODEL ONE QUARTER OF THE PLATE
N,1
N,11,5
FILL
NGEN,2,20,1,11,1,,.6
E,1,2,22,21
EGEN,10,1,1
C*** IMPOSE INITIAL VELOCITY = 5000 OVER CENTRAL SECTION ONLY
ITER,1,0,1
TIME,1E-8
D,ALL,ALL
NSEL,X,0,1
D,ALL,UZ,-5E-5
NALL
LWRITE
C*** DEFINE REMAINING TRANSIENT
ITER,100,0,1
TIME,0.001
DDEL,ALL,ALL
C*** DEFINE BOUNDARY CONDITIONS
SYMBC,,1
SYMBC,,2
NSEL,X,5
D,ALL,ALL                              * FIXED AT END
NALL
LWRITE
AFWRITE
FINISH
/INPUT,27
FINISH
/SHOW,PR15,F33,1
/POST26
C*** RETRIEVE DEFLECTION HISTORY
DISP,2,1,UZ
ADD,3,2,,,DISP,,,-1.0                  * REVERSE SIGN ON UZ
PRVAR,3
/GRAPH,GRID,1
/GRAPH,LABX,TIME
/GRAPH,LABY,DISP
PLVAR,3
FINISH
```

5.19 Large Deflection Under Point Load of Hinged Spherical Shell

```
/VERIFY,STIF43-16
/PREP7
/TITLE, STIF43-16, POINT LOAD ON HINGED SPHERICAL SHELL LARGE DEFLECTION
/COM,  ANALYSIS
/COM,  REF. E.N. DVORKIN AND K.J. BATHE, 'A CONTINUUM MECHANICS BASED
/COM,  FOUR-NODE SHELL ELEMENT FOR GENERAL NONLINEAR ANALYSIS', J. OF ENG.
/COM,  COMPUTATIONS, VOL.1, PP.77-88 1984
CSYS,1
K,1,2540,90,-784.9
LOCAL,11,0,784.9,0,0
K,2,,2400,-784.9
KMOV,2,1,2540,999,-784.9,11,0,999,-784.9
K,3,2540,72.0001,0
KMOV,3,11,0,999,0,2,2658.51,999,0
CSYS,2
K,4,2658.51,90
CSYS,1
L,1,2
LOCAL,12,1,,,,,90
L,2,3
CSYS,2
L,3,4
L,1,4
ET,1,43,,,0
EX,1,68.95
NUXY,1,.3
R,1,99.45
KAY,9,1
KAY,6,1
A,1,2,3,4
ELSI,,3
AMESH,1
WSORT,X
NRSEL,NODE,8,12
SYMBC,2,3
SYMBC,2,2,90
NALL
NRSEL,NODE,1,4
CSYS,1
NROT,ALL
D,ALL,UX,,,,,UY,UZ
D,ALL,ROTX,,,,,ROTZ
NALL
NRSEL,NODE,5,7
CSYS,12
NROT,ALL
D,ALL,UX,,,,,UY,UZ
D,ALL,ROTX,,,,,ROTZ
NALL
CSYS,2
D,2,ALL
```

```
CSYS
ITER,-40
*CREATE,LOAD
D,8,UX,-ARG1
LWRITE
*END
*USE,LOAD,0
RP16,,20
AFWRITE
FINISH
/GOPR
/INPUT,27
FINISH
/SHOW,PR16,F33,1
/POST26
TVAR,1
DISP,2,8,UX
ADD,3,2,,,DISP,,,-1
RFORCE,4,8,FX
ADD,5,4,,,CL,,,-.004
PRVAR,2,4,3,5
/GRAPH,GRID,1
/GRAPH,LABX,DISP
/GRAPH,LABY,LOAD
XVAR,3
PLVAR,5
FINISH
```

5.20 Nonlinear Analysis of a Barrel Vault Roof

```
/VERIFY,STIF43-17
/PREP7
/TITLE, NONLINEAR ANALYSIS OF A BARREL VAULT ROOF
/COM, FROM "NONLINEAR MATERIAL AND GEOMETRIC BEHAVIOR OF SHELL STRUCTURES",
/COM,    G.A.DUPUIS, ET AL, COMPUTERS AND STRUCTURES, VOL.1, P223-239, 1971
ET,1,43
R,1,3
C*** DEFINE MATERIAL PROPERTIES
EX,,3E6
DENS,,.3333333                          * MAKE DENSITY*THICKNESS UNITY
NUXY,,0
KNL,1
NL,,13,10                               * C13=10, BILINEAR KINEMATIC HARDENING
NL,,19,0,10
NL,,25,600,600                          * YIELD=600, PERFECTLY PLASTIC
CSYS,1
K,1,300,90                              * RADIUS=300
K,2,300,50                              * 40 DEGREE ARC
KGEN,2,1,2,1,,,-300                     * HALF LENGTH IS 300
A,1,2,4,3
ELSI,,4                                 * 4 ELEMENTS ON A SIDE
AMESH,1
```

```
WSORT,Z
SYMBC,,1
SYMBC,,3,-300
NSEL,Z,0
D,ALL,UX,,,,,UY                          * DIAPHRAM SUPPORT
NALL
ITER,-20,0,20
POSTR,20,1,5                             * PUT STRAIN INFO ON POST DATA FILE
*CREATE,LOAD
ACEL,,ARG1
LWRITE
*END
*USE,LOAD,0
RP29,,.015                               * LOAD TO 0.42
AFWRITE
FINISH
/INPUT,27
FINISH
/SHOW,PR17,F33,1
/POST26
C*** PLOT LOAD-DEFLECTION CURVE
TVAR,1
DISP,2,6,UY                              * GET VERTICAL DISP AT CENTER OF FREE EDGE
FILL,3,1,29,1,0,.015                     * CREATE TABLE OF ACEL VALUES
ADD,4,2,,,DISP,,,-1.0                    * REVERSE SIGN ON DISP
PRVAR,3,4
/GRAPH,GRID,1
/GRAPH,LABX,DISP
/GRAPH,LABY,LOAD
XVAR,4
PLVAR,3
FINISH
/PREP7
/TITLE, NONLINEAR ANALYSIS OF A BARREL VAULT ROOF
C*** FROM "NONLINEAR MATERIAL AND GEOMETRIC BEHAVIOR OF SHELL STRUCTURES",
C***    G.A.DUPUIS, ET AL, COMPUTERS AND STRUCTURES, VOL.1, P223-239, 1971
KAY,6,1                                  * LARGE DEFLECTION
ET,1,43
R,1,3
C*** DEFINE MATERIAL PROPERTIES
EX,,3E6
DENS,,.3333333                           * MAKE DENSITY*THICKNESS UNITY
NUXY,,0
KNL,1
NL,,13,10                                * C13=10, BILINEAR KINEMATIC HARDENING
NL,,19,0,10
NL,,25,600,600                           * YIELD=600, PERFECTLY PLASTIC
CSYS,1
K,1,300,90                               * RADIUS=300
K,2,300,50                               * 40 DEGREE ARC
KGEN,2,1,2,1,,,-300                      * HALF LENGTH IS 300
A,1,2,4,3
ELSI,,4                                  * 4 ELEMENTS ON A SIDE
```

```
AMESH,1
WSORT,Z
SYMBC,,1
SYMBC,,3,-300
NSEL,Z,0
D,ALL,UX,,,,,UY                          * DIAPHRAM SUPPORT
NALL
ITER,-20,0,20
*CREATE,LOAD
ACEL,,ARG1
LWRITE
*END
*USE,LOAD,0
RP16,,.03                                * LOAD TO 0.45
AFWRITE
FINISH
/INPUT,27
FINISH
/POST26
/NOER
C*** PLOT LOAD-DEFLECTION CURVE
TVAR,1
DISP,2,6,UY                              * GET VERTICAL DISP AT CENTER OF FREE EDGE
FILL,3,1,16,1,0,.03                      * CREATE TABLE OF ACEL VALUES
ADD,4,2,,,DISP,,,-1.0                    * REVERSE SIGN ON DISP
PRVAR,3,4
/GRAPH,GRID,1
/GRAPH,LABX,DISP
/GRAPH,LABY,LOAD
XVAR,4
XRANGE,0,10
PLVAR,3
FINISH
```

6. Solid Modeling Contact Analysis (Model 1)

6.1 Pinion Spur Gear

```
/PREP7                          * ENTER PREPROCESSOR TO BUILD SOLID MODEL
/TITLE, PINION SPUR GEAR
/SHOW,4336M                     * SET DISPLAY TYPE TO TEKTRONIX 4336M
/GET,16,G16T16                  * ASSIGN FILE G16T16 TO UNIT NUMBER 16
/GET,28,G16T28                  * ASSIGN FILE G16T28 TO UNIT NUMBER 28
/COM
/COM                            GEAR TOOTH PROFILE GENERATION
D = 2                           * PITCH DIAMETER
N = 16                          * NUMBER OF TEETH
CSYS,1                          * CYLINDRICAL COORDINATE SYSTEM
*USE,GTOOTH,D,N                 * GENERATE GEAR TOOTH PROFILE DEFINING
/COM                               RDBC, RDAC, RDRC, TANG
/PNUM,KPOI,1                    * TURN ON KEYPOINT NUMBER DISPLAY
LPLOT                           * PLOT GEAR TOOTH PROFILE
/PNUM,KPOI,0                    * TURN OFF KEYPOINT NUMBER DISPLAY
/COM
/COM                            GENERATE SOLID MODEL OF ONE HALF GEAR TOOTH
RDC = RDRC*0.7                  * RADIUS OF CENTRAL REGION
K,6,RDC,0                       * KEYPOINT ON RADIAL LINE TO CENTER OF TOP LAND
L,5,6                           * RADIAL LINE THROUGH CENTER OF TOP LAND
K,7,RDC,-TANG                   * KEYPT ON RADIAL LINE TO CENTER OF BOTTOM LAND
L,7,3                           * RADIAL LINE THROUGH CENTER OF BOTTOM LAND
L,6,7                           * ARC DEFINING INNER RADIUS OF TOOTH SEGMENT
LDVS,ALL,RDRC*0.2               * SET ELEMENT SIZE ON ALL LINES
AL,ALL                          * AREA BOUNDED BY ALL (7) LINES
/COM
/COM                            MESH HALF TOOTH
ET,1,82                         * ELEMENT TYPE 1, ID NO. 82, 2D 8 NODE ISO SOLID
TYPE,1                          * ELEMENTS TO BE GENERATED NEXT ARE 2D SOLIDS
MAT,1                           * ELEMENTS TO BE GENERATED ARE MATERIAL 1
ELSIZE,,,,1                     * STRAIGHT-SIDED ELEMENTS
AMESH,1                         * MESH HALF TOOTH
/COM
/COM                            GENERATE OTHER HALF TOOTH BY SYMMETRY
ARSYM,2,ALL                     * REFLECT HALF TOOTH AREA ABOUT THETA=0
EPLOT                           * PLOT ELEMENTS IN SINGLE TOOTH
/COM
/COM                            GENERATE REST OF TEETH BY ROTATION
AGEN,N,ALL,,,,TANG*2            * GENERATE REST OF TEETH BY ROTATING FIRST TOOTH
NUMMRG,ALL                      * MERGE COINCIDENT KEYPOINTS, LINES, AND NODES
LPLOT                           * PLOT OUTER RING OF TEETH
/COM
/COM                            GENERATE SOLID MODEL OF CENTRAL REGION GEOMETRY
RDH = 0.4                       * RADIUS OF CENTER OF HOLE IN GEAR BLANK
RH = 0.1                        * RADIUS OF HOLE
RDHB = 0.2                      * RADIUS OF HUB
KPINV                           * DESELECT ALL KEYPOINTS
LSINV                           * DESELECT ALL LINES
```

```
K,,RDH,0                    * KEYPOINT AT CENTER OF HOLE
KGEN,2,11,,,,,1             * KEYPOINT COMPLETING DEFINITION OF HOLE AXIS
CIRCLE,11,RH,12,6           * GENERATE CIRCUMFERENCE OF HOLE 1
K,,0,0                      * KEYPOINT AT CENTER OF HUB
KGEN,2,30,,,,,1             * KEYPOINT COMPLETING DEFINITION OF HUB AXIS
CIRCLE,30,RDHB,31,6         * GENERATE CIRCUMFERENCE OF HUB
LDVS,ALL,,2                 * SET NUMBER OF ELEMENT DIVISIONS PER LINE TO 2
KPALL                       * RESTORE ALL KEYPOINTS
LSALL                       * RESTORE ALL LINES
/PNUM,KPOI,1                * TURN ON KEYPOINT NUMBER DISPLAY
LPLOT                       * PLOT CENTRAL REGION GEOMETRY AND TEETH
/PNUM,KPOI,0                * TURN OFF KEYPOINT NUMBER DISPLAY
/COM
/COM                        DIVIDE REGION INTO 2 SIMPLY CONNECTED AREAS
CSYS,0                      * CARTESIAN COORDINATE SYSTEM
L,6,16                      * LINE BETWEEN HOLE 1 AND REGION BOUNDARY
L,40,26                     * LINE BETWEEN HUB AND HOLE 1
L,44,116                    * LINE BETWEEN HUB AND REGION BOUNDARY
LDVS,ALL,RDRC*0.1           * SET ELEMENT SIZE ON ALL LINES
/COM
CSYS,1                      * CYLINDRICAL COORDINATE SYSTEM
KPSEL,X,0,RDC*1.001         * SELECT KEYPOINTS IN CENTRAL REGION
CSYS,0                      * CARTESIAN COORDINATE SYSTEM
KPRSEL,Y,RH*(-0.1),RDAC     * RESELECT KEYPOINTS ON X AXIS AND ABOVE
LSKP,1                      * SELECT LINES CONNECTED TO THESE KEYPOINTS
AL,ALL                      * DEFINE TOP (+Y) AREA IN CENTRAL REGION
/COM
/COM                        MESH CENTRAL REGION
KPALL                       * RESTORE ALL KEYPOINTS
LSALL                       * RESTORE ALL LINES
AMESH,ALL                   * MESH TOP AREA IN CENTRAL REGION
ARSYM,2,(N*2)+1             * REFLECT TOP AREA ABOUT Y=0
NUMMRG,ALL                  * MERGE COINCIDENT KEYPOINTS, LINES, AND NODES
EPLOT                       * PLOT (COARSE) MESH OF ENTIRE GEAR
/COM
/COM                        REORDER AND DEFINE MATERIAL PROPERTIES
EX,1,3E7                    * YOUNG'S MODULUS (PSI)
DENS,1,0.00073              * MASS DENSITY (LBF SEC**2 / IN**4)
R,2,1,0                     * (DUMMY) REAL CONSTANTS FOR SPARS
EX,2,1                      * (DUMMY) MODULUS FOR SPARS
NUMCMP,ALL                  * COMPRESS THE DATABASE
CDWRITE                     * WRITE DATA TO CODED FILE G16T28
FINISH                      * END PINION SOLID MODEL GENERATION
/COM
/EOF                        * END MODEL GENERATION RUN
```

6.2 Driven Spur Gear (Model 2)

```
/PREP7                          * ENTER PREPROCESSOR TO BUILD SOLID MODEL
/TITLE, DRIVEN SPUR GEAR
/SHOW,4336M                     * SET DISPLAY TYPE TO TEKTRONIX 4336M
/GET,16,G40T16                  * ASSIGN FILE G40T16 TO UNIT NUMBER 16
/GET,28,G40T28                  * ASSIGN FILE G40T28 TO UNIT NUMBER 28
/COM
/COM                            GEAR TOOTH PROFILE GENERATION
D = 5                           * PITCH DIAMETER
N = 40                          * NUMBER OF TEETH
CSYS,1                          * CYLINDRICAL COORDINATE SYSTEM
*USE,GTOOTH,D,N                 * GENERATE GEAR TOOTH PROFILE DEFINING
/COM                               RDBC, RDAC, RDRC, TANG
/PNUM,KPOI,1                    * TURN ON KEYPOINT NUMBER DISPLAY
LPLOT                           * PLOT GEAR TOOTH PROFILE
/PNUM,KPOI,0                    * TURN OFF KEYPOINT NUMBER DISPLAY
/COM
/COM                            GENERATE SOLID MODEL OF ONE HALF GEAR TOOTH
RDC = RDRC*0.9                  * RADIUS OF CENTRAL REGION
K,6,RDC,0                       * KEYPOINT ON RADIAL LINE TO CENTER OF TOP LAND
L,5,6                           * RADIAL LINE THROUGH CENTER OF TOP LAND
K,7,RDC,-TANG                   * KEYPT ON RADIAL LINE TO CENTER OF BOTTOM LAND
L,7,3                           * RADIAL LINE THROUGH CENTER OF BOTTOM LAND
L,6,7                           * ARC DEFINING INNER RADIUS OF TOOTH SEGMENT
LDVS,ALL,RDRC*0.08              * SET ELEMENT SIZE ON ALL LINES
AL,ALL                          * AREA BOUNDED BY ALL (7) LINES
/COM
/COM                            MESH HALF TOOTH
ET,1,82                         * ELEMENT TYPE 1, ID NO. 82, 2D 8 NODE ISO SOLID
TYPE,1                          * ELEMENTS TO BE GENERATED NEXT ARE 2D SOLIDS
MAT,2                           * ELEMENTS TO BE GENERATED ARE MATERIAL 2
ELSIZE,,,,1                     * STRAIGHT-SIDED ELEMENTS
AMESH,1                         * MESH HALF TOOTH
/COM
/COM                            GENERATE OTHER HALF TOOTH BY SYMMETRY
ARSYM,2,ALL                     * REFLECT HALF TOOTH AREA ABOUT THETA=0
EPLOT                           * PLOT ELEMENTS IN SINGLE TOOTH
/COM
/COM                            GENERATE REST OF TEETH BY ROTATION
AGEN,N,ALL,,,,TANG*2            * GENERATE REST OF TEETH BY ROTATING FIRST TOOTH
NUMMRG,ALL                      * MERGE COINCIDENT KEYPOINTS, LINES, AND NODES
LPLOT                           * PLOT OUTER RING OF TEETH
/COM
/COM                            GENERATE SOLID MODEL OF CENTRAL REGION GEOMETRY
RDH = 1.3                       * RADIUS OF CENTERS OF 5 HOLES IN GEAR BLANK
RH = 0.5                        * RADIUS OF HOLES
RDHB = 0.4                      * RADIUS OF HUB
KPINV                           * DESELECT ALL KEYPOINTS
LSINV                           * DESELECT ALL LINES
K,,RDH,0                        * KEYPOINT AT CENTER OF HOLE
KGEN,2,11,,,,,1                 * KEYPOINT COMPLETING DEFINITION OF HOLE AXIS
CIRCLE,11,RH,12,6               * GENERATE CIRCUMFERENCE OF HOLE 1
```

```
LGEN,5,ALL,,,,(360/5)          * GENERATE CIRCUMFERENCES OF OTHER 4 HOLES
K,,0,0                         * KEYPOINT AT CENTER OF HUB
KGEN,2,86,,,,,1                * KEYPOINT COMPLETING DEFINITION OF HUB AXIS
CIRCLE,86,RDHB,87,6            * GENERATE CIRCUMFERENCE OF HUB
LDVS,ALL,,2                    * SET NUMBER OF ELEMENT DIVISIONS PER LINE TO 2
KPALL                          * RESTORE ALL KEYPOINTS
LSALL                          * RESTORE ALL LINES
/PNUM,KPOI,1                   * TURN ON KEYPOINT NUMBER DISPLAY
LPLOT                          * PLOT CENTRAL REGION GEOMETRY AND TEETH
/PNUM,KPOI,0                   * TURN OFF KEYPOINT NUMBER DISPLAY
/COM
/COM                           DIVIDE REGION INTO 5 SIMPLY CONNECTED AREAS
CSYS,0                         * CARTESIAN COORDINATE SYSTEM
L,6,16                         * LINE BETWEEN HOLE 1 AND REGION BOUNDARY
L,17,41                        * LINE BETWEEN HOLES 1 AND 2
L,31,55                        * LINE BETWEEN HOLES 2 AND 3
L,44,228                       * LINE BETWEEN HOLE 3 AND REGION BOUNDARY
L,45,69                        * LINE BETWEEN HOLES 3 AND 4
KPSEL,POIN,45                  * SELECT KEYPOINT AT ONE END OF LINE
KPASEL,POIN,69                 * ADD KEYPOINT AT OTHER END OF LINE
LSKP,1                         * SELECT LINE CONNECTED TO THESE KEYPOINTS
*GET,LS34,LSMX                 * GET LINE NUMBER
LDIV,LS34                      * DIVIDE LINE INTO 2 EQUAL LINES
KPALL                          * RESTORE ALL KEYPOINTS
LSALL                          * RESTORE ALL LINES
L,58,340                       * LINE BETWEEN HOLE 4 AND REGION BOUNDARY
L,59,83                        * LINE BETWEEN HOLES 4 AND 5
L,73,27                        * LINE BETWEEN HOLES 5 AND 1
L,96,26                        * LINE BETWEEN HUB AND HOLE 1
L,100,110                      * LINE BETWEEN HUB AND HOLE 3-4 LINE
LDVS,ALL,RDRC*0.1              * SET ELEMENT SIZE ON ALL LINES
/COM
CSYS,1                         * CYLINDRICAL COORDINATE SYSTEM
KPSEL,X,RDH,RDC*1.001          * SELECT KEYPTS BETWEEN HOLE CENTERS AND BDRY
TH1 = 0                        * STARTING ANGLE
TH2 = ((2)*360)/5              * ENDING ANGLE
KPRSEL,Y,TH1-0.1,TH2+0.1       * RESELECT KEYPOINTS BETWEEN ANGLES
LSKP,1                         * SELECT LINES CONNECTED TO THESE KEYPOINTS
AL,ALL                         * DEFINE 1ST AREA IN CENTRAL REGION
/COM
KPSEL,X,RDH,RDC*1.001          * SELECT KEYPTS BETWEEN HOLE CENTERS AND BDRY
TH1 = TH2                      * STARTING ANGLE
TH2 = ((3)*360)/5              * ENDING ANGLE
CSCIR,1,1                      * CHANGE SINGULARITY FROM 180 DEG TO 0 DEG
KPRSEL,Y,TH1-0.1,TH2+0.1       * RESELECT KEYPOINTS BETWEEN ANGLES
CSCIR,1,0                      * CHANGE SINGULARITY FROM 0 DEG TO 180 DEG
LSKP,1                         * SELECT LINES CONNECTED TO THESE KEYPOINTS
AL,ALL                         * DEFINE 2ND AREA IN CENTRAL REGION
/COM
KPSEL,X,0,RDH+(RH*0.9)         * SELECT KEYPOINTS IN INNER PART OF REGION
CSYS,0                         * CARTESIAN COORDINATE SYSTEM
KPRSEL,Y,-0.1,RDC              * RESELECT KEYPOINTS ON X AXIS AND ABOVE
LSKP,1                         * SELECT LINES CONNECTED TO THESE KEYPOINTS
```

```
AL,ALL                          * DEFINE 3RD AREA IN CENTRAL REGION
/COM
/COM                            MESH CENTRAL REGION
KPALL                           * RESTORE ALL KEYPOINTS
LSALL                           * RESTORE ALL LINES
AMESH,ALL                       * MESH 3 AREAS IN CENTRAL REGION
ARSYM,2,(N*2)+1,(N*2)+3,2       * REFLECT 1ST AND 3RD AREA ABOUT Y=0
NUMMRG,ALL                      * MERGE COINCIDENT KEYPOINTS, LINES, AND NODES
EPLOT                           * PLOT (COARSE) MESH OF ENTIRE GEAR
/COM
/COM                            DISPLAY CLOSE-UP OF GEAR TOOTH CONTACT AREA
ARSEL,AREA,1,4                  * SELECT 2 TEETH
ARASEL,AREA,(N*2)-1,N*2         * SELECT 3RD TOOTH
LSAR,1                          * SELECT LINES IN AREAS
KPLS                            * SELECT KEYPOINTS ON LINES
NAREA,1                         * SELECT NODES IN AREAS
EAREA                           * SELECT ELEMENTS IN AREAS
/PNUM,LINE,1                    * TURN ON LINE NUMBER DISPLAY
LPLOT                           * PLOT CONTACT AREA
/PNUM,LINE,0                    * TURN OFF LINE NUMBER DISPLAY
/COM
/COM                            MESH LINES WITH DUMMY SPARS FOR NICE NUMBERING
ACLEAR,(N*2)-1,N*2              * CLEAR COARSE MESH FROM 1 TOOTH (2 HALVES)
ACLEAR,1,4                      * CLEAR COARSE MESH FROM 2 MORE TEETH
LDVS,ALL,0.05,,,1               * SET SMALLER ELEMENT SIZE ON ALL UNMESHED LINES
LSSEL,LINE,13                   * SELECT 1ST LINE TO BE CONNECTED TO GAP ELTS
LSASEL,LINE,24                  * SELECT 2ND LINE TO BE CONNECTED TO GAP ELTS
LSASEL,LINE,556                 * SELECT 3RD LINE TO BE CONNECTED TO GAP ELTS
ET,2,1                          * ELEMENT TYPE 2, ID NO. 1, 2D SPAR
TYPE,2                          * ELEMENTS TO BE GENERATED NEXT ARE SPARS
REAL,3                          * ELEMENTS TO BE GEN NEXT ARE REAL CONST SET 3
MAT,3                           * ELEMENTS TO BE GEN NEXT ARE MATERIAL SET 3
LMESH,ALL                       * MESH SELECTED LINES
/COM
/COM                            MESH AREAS WITH 2D SOLID ELEMENTS
TYPE,1                          * ELEMENTS TO BE GENERATED NEXT ARE 2D SOLIDS
ELSIZE,,,0,0                    * CURVED QUADRILATERAL OR TRIANGULAR ELEMENTS
AMESH,4                         * MESH HALF TOOTH IN FIRST TRANSITION AREA
AMESH,3                         * MESH HALF TOOTH WITH REFINED MESH
AMESH,2                         * MESH HALF TOOTH WITH REFINED MESH
ET,2,0                          * DEACTIVATE DUMMY SPAR ELEMENTS
CSYS,1                          * CYLINDRICAL COORDINATE SYSTEM
AGEN,2,2,3,,,-TANG*2            * GENERATE REMAINING TOOTH BY ROTATING
AMESH,(N*2)-1                   * MESH HALF TOOTH IN SECOND TRANSITION AREA
ET,2,1                          * ACTIVATE DUMMY SPAR ELEMENTS
NUMMRG,ALL                      * MERGE COINCIDENT KEYPTS, LINES, NODES, AREAS
EPLOT                           * PLOT REFINED MESH IN CONTACT AREA
/COM
/COM                            RESTORE ENTIRE MODEL
ARSEL,,1,100,,1                 * SELECT KEYPOINTS, ETC., ASSOCIATED WITH AREAS
KPINV                           * SELECT KEYPOINTS NOT ASSOCIATED WITH AREAS
KDELE,ALL                       * DELETE THESE CONSTRUCTION KEYPOINTS
KPALL                           * RESTORE ALL KEYPOINTS
```

```
EALL                                * RESTORE ALL ELEMENTS
EPLOT                               * PLOT (COARSE AND FINE) MESH OF ENTIRE GEAR
/COM
/COM                                REORDER AND DEFINE MATERIAL PROPERTIES
WFRONT                              * ESTIMATE CURRENT MAX WAVEFRONT
CSYS,0                              * CARTESIAN COORDINATE SYSTEM
WSORT,Y                             * REORDER ELEMENTS TO REDUCE WAVEFRONT
EX,2,3E7                            * YOUNG'S MODULUS (PSI)
DENS,2,0.00073                      * MASS DENSITY (LBF SEC**2 / IN**4)
R,3,1,0                             * (DUMMY) REAL CONSTANTS FOR SPARS
EX,3,1                              * (DUMMY) MODULUS FOR SPARS
NUMCMP,ALL                          * COMPRESS THE DATABASE
CDWRITE                             * WRITE DATA TO CODED FILE G40T28
FINISH                              * END DRIVEN GEAR SOLID MODEL DEFINITION
/COM
/EOF                                * END MODEL GENERATION RUN
```

6.3 Spur Gear Combine Model 1 and Model 2

```
/PREP7                              * ENTER PREPROCESSOR TO COMBINE MODELS
/SHOW,GCOM,F33,1                    * PUT VECTOR MODE DISPLAYS ON FILE GCOM.F33
/GET,16,GCOMT16                     * ASSIGN FILE GCOMT16 TO UNIT NUMBER 16
/COM
/COM                                READ IN FIRST MODEL, GEAR 1
/INP,G16T28                         * READ FILE G16T28 CONTAINING 1ST MODEL
/INP,G16PARM                        * READ PARAMETERS FROM 1ST MODEL
/TITLE, COMBINING TWO SOLID MODELS
/COM
/COM                                SET PARAMETERS FOR LATER USE
D1 = D                              * PITCH DIAMETER OF GEAR 1
N1 = N                              * NUMBER OF TEETH FOR GEAR 1
ANG1 = TANG                         * TOOTH ANGLE FOR GEAR 1
HB1 = RDHB                          * RADIUS OF HUB FOR GEAR 1
D2 = 5                              * PITCH DIAMETER OF GEAR 2
/COM
/COM                                TRANSLATE AND ROTATE TO NEW POSITION
NUMCMP,ALL                          * COMPRESS THE DATABASE
*GET,AMX1,ARMX                      * GET MAX AREA NUMBER OF ORIGINAL GEAR 1
LOCAL,30,0,(D1+D2)/2,0,0,ANG1       * ROTATED CARTESIAN CS AT NEW CENTER OF GEAR 1
CSYS,0                              * CARTESIAN COORDINATE SYSTEM
ATRAN,30,ALL                        * ROTATE AND TRANSLATE COPY OF GEAR 1
ARSEL,AREA,1,AMX1,,1                * SELECT ORIGINAL GEAR 1 AREAS
ACLEAR,ALL                          * CLEAR MESH FROM ORIGINAL GEAR 1 AREAS
LCLEAR,ALL                          * CLEAR MESH FROM ORIGINAL GEAR 1 LINES
ADELE,ALL,,,1                       * DELETE ORIGINAL GEAR 1 AREAS, LINES, KEYPOINTS
KPALL                               * RESTORE ALL KEYPOINTS
LSALL                               * RESTORE ALL LINES
ARALL                               * RESTORE ALL AREAS
NALL                                * RESTORE ALL NODES
EALL                                * RESTORE ALL ELEMENTS
NUMCMP,ALL                          * COMPRESS THE DATABASE
LPLOT                               * PLOT TRANSLATED AND ROTATED GEAR 1
```

```
/COM
/COM                                READ IN SECOND MODEL, GEAR 2
/INP,G40T28                         * READ FILE G40T28 CONTAINING 2ND MODEL
/INP,G40PARM                        * READ PARAMETERS FROM 2ND MODEL
/TITLE, COMBINING TWO SOLID MODELS
/COM
/COM                                SET PARAMETERS FOR LATER USE
N2 = N                              * NUMBER OF TEETH FOR GEAR 2
ANG2 = TANG                         * TOOTH ANGLE FOR GEAR 2
HB2 = RDHB                          * RADIUS OF HUB FOR GEAR 2
D =                                 * DELETE DUPLICATE PARAMETER
N =                                 * DELETE DUPLICATE PARAMETER
TANG =                              * DELETE DUPLICATE PARAMETER
RDHB =                              * DELETE DUPLICATE PARAMETER
/COM
NUMMRG,TYPE                         * MERGE COINCIDENT ELEMENT TYPES
EPLOT                               * PLOT MESH OF BOTH GEARS
FINISH                              * END COMBINING MODELS
/COM
/EOF                                * END COMBINING MODELS RUN
```

6.4 Generation Pass Spur Gear

```
/COM   -------------------------GENERATION PASS---------------------------------
/COM
/COM                                RENAME RESUME FILE FOR BACKUP
/GET,30,GCOMT16                     * ASSIGN FILE GCOMT16 TO UNIT NUMBER 30
/GET,2,GGENT2                       * ASSIGN FILE GGENT2 TO UNIT NUMBER 2
/GET,8,GGENT8                       * ASSIGN FILE GGENT8 TO UNIT NUMBER 8
/GET,11,GGENT11                     * ASSIGN FILE GGENT11 TO UNIT NUMBER 11
/GET,15,GGENT15                     * ASSIGN FILE GGENT15 TO UNIT NUMBER 15
/GET,16,GGENT16                     * ASSIGN FILE GGENT16 TO UNIT NUMBER 16
/COPY,30,16                         * COPY FILE30=GCOMT16 TO FILE16=GGENT16
/COM
/PREP7                              * ENTER PREPROCESSOR TO GENERATE SUBSTRUCTURE
/SHOW,4336M                         * SET DISPLAY TYPE TO TEKTRONIX 4336M
RESUME                              * RESUME FROM "COMBINING TWO MODELS" RUN
/TITLE, SUBSTRUCTURE GENERATION PASS
/COM
/COM                                REFINE CONTACT AREA OF GEAR 1; FIRST
/COM                                 DISPLAY CLOSE-UP OF GEAR TOOTH CONTACT AREA
AR1 = ((N2*2)+5)+(N1-2)             * STARTING AREA NUMBER IN GEAR 1 CONTACT AREA
ARSEL,AREA,AR1,AR1+5,,1             * SELECT 3 TEETH: KEYPTS,LINES,AREAS,NODES,ELTS
/PNUM,LINE,1                        * TURN ON LINE NUMBER DISPLAY
LPLOT                               * PLOT CONTACT AREA
/PNUM,LINE,0                        * TURN OFF LINE NUMBER DISPLAY
/COM
/COM                                MESH LINES WITH DUMMY SPARS FOR NICE NUMBERING
ACLEAR,ALL                          * CLEAR COARSE MESH FROM 3 TOOTH AREA
LDVS,ALL,0.05,,,1                   * SET SMALLER ELEMENT SIZE ON ALL UNMESHED LINES
LSSEL,LINE,597,621,12               * SELECT 3 LINES TO BE CONNECTED TO GAP ELTS
ET,2,1                              * ELEMENT TYPE 2, ID NO. 1, 2D SPAR
```

```
TYPE,2                          * ELEMENTS TO BE GENERATED NEXT ARE SPARS
REAL,2                          * ELEMENTS TO BE GEN NEXT ARE REAL CONST SET 2
MAT,2                           * ELEMENTS TO BE GEN NEXT ARE MATERIAL SET 2
LMESH,ALL                       * MESH SELECTED LINES
/COM
/COM                            MESH AREAS WITH 2D SOLID ELEMENTS
TYPE,1                          * ELEMENTS TO BE GENERATED NEXT ARE 2D SOLIDS
ELSIZE,,,0,0                    * CURVED QUADRILATERAL OR TRIANGULAR ELEMENTS
AMESH,AR1                       * MESH HALF TOOTH IN SECOND TRANSITION AREA
AMESH,AR1+1                     * MESH HALF TOOTH WITH REFINED MESH
AMESH,AR1+2                     * MESH HALF TOOTH WITH REFINED MESH
ET,2,0                          * DEACTIVATE DUMMY SPAR ELEMENTS
LOCAL,31,1,(D1+D2)/2,0,0,ANG1   * CYLINDRICAL CS AT CENTER OF GEAR 1
AGEN,2,AR1+1,AR1+2,,,ANG1*2     * GENERATE REMAINING TOOTH BY ROTATING
ELSIZE,,,1,0                    * CURVED TRIANGULAR ELEMENTS (TRI-QUAD FAILS)
AMESH,AR1+5                     * MESH HALF TOOTH IN FIRST TRANSITION AREA
EPLOT                           * PLOT REFINED MESH IN CONTACT AREA
/COM
/COM                            REORDER GEAR 1
ARSEL,,(N2*2)+6,1000,,1         * SELECT AREAS IN GEAR 1
NUMMRG,ALL                      * MERGE COINCIDENT KEYPTS, LINES, NODES, AREAS
CSYS,0                          * CARTESIAN COORDINATE SYSTEM
WFRONT                          * ESTIMATE CURRENT MAX WAVEFRONT
WSORT,Y                         * REORDER ELEMENTS TO REDUCE WAVEFRONT
/COM
/COM                            RESTORE ENTIRE MODEL
ARSEL,,1,1000,,1                * SELECT KEYPOINTS, ETC., ASSOCIATED WITH AREAS
KPINV                           * SELECT KEYPOINTS NOT ASSOCIATED WITH AREAS
KDELE,ALL                       * DELETE THESE CONSTRUCTION KEYPOINTS
KPALL                           * RESTORE ALL KEYPOINTS
EALL                            * RESTORE ALL ELEMENTS
NUMCMP,ALL                      * COMPRESS THE DATABASE
EPLOT                           * PLOT (COARSE AND FINE) MESH OF BOTH GEARS
/COM
/COM                            ROTATE NODAL COORD SYSTEMS TO EACH GEAR'S CYL CS
CSYS,31                         * CYLINDRICAL CS AT CENTER OF GEAR 1
ARSEL,,(N2*2)+6,1000,,1         * SELECT AREAS IN GEAR 1
NROTAT,ALL                      * ROTATE NODAL COORDINATE SYSTEMS OF GEAR 1
CSYS,1                          * CYLINDRICAL CS AT CENTER OF GEAR 2
ARSEL,,1,(N2*2)+5,,1            * SELECT AREAS IN GEAR 2
NROTAT,ALL                      * ROTATE NODAL COORDINATE SYSTEMS OF GEAR 2
LSALL                           * RESTORE ALL LINES
ARALL                           * RESTORE ALL AREAS
/COM
/COM                            DEFINE MASTER DOFS AT BDRY OF 3 TEETH
ET,2,1                          * REACTIVATE DUMMY SPAR ELEMENTS
ESEL,TYPE,2                     * SELECT SPAR ELEMENTS
NELEM                           * SELECT NODES IN SELECTED ELEMENTS
M,ALL,ALL                       * DEFINE DOFS AT SELECTED NODES AS MASTER DOFS
NWRITE                          * WRITE MASTER DOF NODE DATA TO FILE15=GGENT15
ET,2,0                          * DEACTIVATE DUMMY SPAR ELEMENTS
EALL                            * RESTORE ALL ELEMENTS
/PBC,ALL,1                      * TURN ON DISPLAY OF ALL BOUNDARY CONDITIONS
```

```
NPLOT                   * PLOT CONTACT NODES, SHOWING MASTER DOFS
/COM
/COM                    DEFINE 4 MORE MASTER DOFS AT HUBS
CSYS,31                 * CYLINDRICAL CS AT CENTER OF GEAR 1
KPSEL,X,HB1             * SELECT KEYPOINTS ON CIRCUMFERENCE OF HUB 1
KPRSEL,Y,0              * RESELECT KEYPOINT AT THETA = 0
NKPOI                   * SELECT NODE AT THIS KEYPOINT
M,ALL,UY                * DEFINE THETA DISPL DOF AT NODE AS MASTER DOF
NWRITE                  * WRITE MASTER DOF NODE DATA TO FILE15=GGENT15
/COM
KPSEL,X,HB1             * SELECT KEYPOINTS ON CIRCUMFERENCE OF HUB 1
KPRSEL,Y,180            * RESELECT KEYPOINT AT THETA = 180
NKPOI                   * SELECT NODE AT THIS KEYPOINT
M,ALL,UY                * DEFINE THETA DISPL DOF AT NODE AS MASTER DOF
NWRITE                  * WRITE MASTER DOF NODE DATA TO FILE15=GGENT15
/COM
CSYS,1                  * CYLINDRICAL CS AT CENTER OF GEAR 2
KPSEL,X,HB2             * SELECT KEYPOINTS ON CIRCUMFERENCE OF HUB 2
KPRSEL,Y,0              * RESELECT KEYPOINT AT THETA = 0
NKPOI                   * SELECT NODE AT THIS KEYPOINT
M,ALL,UY                * DEFINE THETA DISPL DOF AT NODE AS MASTER DOF
NWRITE                  * WRITE MASTER DOF NODE DATA TO FILE15=GGENT15
/COM
KPSEL,X,HB2             * SELECT KEYPOINTS ON CIRCUMFERENCE OF HUB 2
KPRSEL,Y,180            * RESELECT KEYPOINT AT THETA = 180
NKPOI                   * SELECT NODE AT THIS KEYPOINT
M,ALL,UY                * DEFINE THETA DISPL DOF AT NODE AS MASTER DOF
NWRITE                  * WRITE MASTER DOF NODE DATA TO FILE15=GGENT15
KPALL                   * RESTORE ALL KEYPOINTS
NALL                    * RESTORE ALL NODES
/COM
/COM                    SET BOUNDARY CONDITIONS ON HUBS
CSYS,31                 * CYLINDRICAL CS AT CENTER OF GEAR 1
KPSEL,X,HB1             * SELECT KEYPOINTS ON CIRCUMFERENCE OF HUB 1
KD,ALL,UX,,,1           * SET R DISPL AT HUB 1 KEYPTS AND NODES TO ZERO
CSYS,1                  * CYLINDRICAL CS AT CENTER OF GEAR 2
KPSEL,X,HB2             * SELECT KEYPOINTS ON CIRCUMFERENCE OF HUB 2
KD,ALL,UX,,,1           * SET R DISPL AT HUB 2 KEYPTS AND NODES TO ZERO
KPALL                   * RESTORE ALL KEYPOINTS
SBCTRA                  * TRANSFER KEYPOINT BCS TO NODES TO CHECK
NPLOT                   * PLOT NODES, SHOWING MASTER DOFS AND DISPL BCS
/COM
/COM                    DEFINE ANALYSIS TYPE AND WRITE FILE
KAN,7                   * SUBSTRUCTURE ANALYSIS
AFWRITE                 * WRITE ANALYSIS FILE (FILE27)
FINISH                  * END MODEL DEFINITION
/COM
/INPUT,27               * BEGIN SOLUTION, READ ANALYSIS FILE
FINISH                  * END SOLUTION
/COM
/EOF                    * END SUBSTRUCTURE GENERATION PASS
```

6.5 Use Pass Gear Model

```
/COM   -------------------------USE PASS-----------------------------------
/COM
/COM                            BACK UP FILES 2 AND 11 FROM THE GENERATION
/COM                             PASS BEFORE RUNNING THIS USE PASS
/COM
/PREP7                          * ENTER PREPROCESSOR TO DEFINE MODEL
/TITLE, SUBSTRUCTURE USE PASS
/SHOW,4336M                     * SET DISPLAY TYPE TO TEKTRONIX 4336M
/GET,8,GGENT8                   * ASSIGN FILE GGENT8 TO UNIT NUMBER 8
/GET,13,GUSET13                 * ASSIGN FILE GUSET13 TO UNIT NUMBER 13
/GET,15,GGENT15                 * ASSIGN FILE GGENT15 TO UNIT NUMBER 15
/GET,16,GUSET16                 * ASSIGN FILE GUSET16 TO UNIT NUMBER 16
/COM
/COM                            DEFINE SUPERELEMENT (= SUBSTRUCTURE)
ET,1,50                         * ELT TYPE 1, ID NO. 50, SUPERELEMENT
E,1,8                           * DEFINE MATRIX NUMBER AND FILE NUMBER
DOF,UX,UY                       * DEFINE DOFS OF SUPERELEMENT
/COM
/COM                            READ IN GAP NODES AND 4 HUB NODES, AND DISPLAY
NREAD                           * READ NODE DATA FROM FILE15
NPLOT                           * PLOT NODES
CSYS,1                          * CYLINDRICAL CS AT CENTER OF DRIVEN GEAR
C1 = (D1+D2)/2                  * DEFINE CENTER OF PINION
OHB1 = HB1*1.1                  * DEFINE OUTSIDE OF PINION HUB (HUB 1)
OHB2 = HB2*1.1                  * DEFINE OUTSIDE OF DRIVEN GEAR HUB (HUB 2)
NSEL,X,OHB2,C1-OHB1             * SELECT GAP NODES
/PNUM,NODE,1                    * TURN ON NODE NUMBER DISPLAY
NPLOT                           * PLOT NODES
/COM
/COM                            DEFINE GAP ELEMENTS
ET,3,52                         * ELT TYPE 3, ID NO. 52, 3D INTERFACE (= GAP)
KEYOPT,3,3,1                    * SMALL STIFFNESS ASSIGNED TO OPEN GAP
KEYOPT,3,4,1                    * DETERMINE GAP SIZE FROM INITIAL NODE LOCATIONS
TYPE,3                          * ELEMENTS TO BE GENERATED NEXT ARE GAPS
REAL,4                          * ELEMENTS TO BE GEN NEXT ARE REAL CONST SET 4
R,4,3E8                         * INTERFACE STIFFNESS
MAT,4                           * ELEMENTS TO BE GEN NEXT ARE MATERIAL SET 4
MU,4,0                          * COEFFICIENT OF FRICTION (MAY BE NONZERO)
/COM
/COM                            GAP ELEMENTS BETWEEN 1ST PAIR OF TEETH
E,4160,2587                     * DEFINE GAP ELT BETWEEN SUPERELEMENT NODES
RP2,1,-1                        * REPEAT PREVIOUS COMMAND TO GEN 1 MORE GAP ELT
E,4162,2592                     * DEFINE GAP ELT BETWEEN SUPERELEMENT NODES
RP5,1,-1                        * REPEAT PREVIOUS COMMAND TO GEN 4 MORE GAP ELTS
/COM
/COM                            GAP ELEMENTS BETWEEN 2ND PAIR OF TEETH
E,4167,2580                     * DEFINE GAP ELT BETWEEN SUPERELEMENT NODES
RP2,1,-1                        * REPEAT PREVIOUS COMMAND TO GEN 1 MORE GAP ELT
E,4169,2585                     * DEFINE GAP ELT BETWEEN SUPERELEMENT NODES
RP5,1,-1                        * REPEAT PREVIOUS COMMAND TO GEN 4 MORE GAP ELTS
/COM
```

```
/COM                                GAP ELEMENTS BETWEEN 3RD PAIR OF TEETH
E,4174,2594                         * DEFINE GAP ELT BETWEEN SUPERELEMENT NODES
RP2,1,-1                            * REPEAT PREVIOUS COMMAND TO GEN 1 MORE GAP ELT
E,4176,2599                         * DEFINE GAP ELT BETWEEN SUPERELEMENT NODES
RP5,1,-1                            * REPEAT PREVIOUS COMMAND TO GEN 4 MORE GAP ELTS
EUSEL,ELEM,1                        * UNSELECT SUPERELEMENT, WHICH WON'T PLOT
EPLOT                               * PLOT GAP ELEMENTS
/COM
/COM                                IMPOSE LOAD OF FORCE COUPLE AT PINION HUB
NSELE,X,C1-OHB1,C1+OHB1             * SELECT NODES ON PINION HUB
F,ALL,FY,400                        * PUT FORCE IN THETA DIRECTION (CCW) ON NODES
NSELE,X,0,OHB2                      * SELECT NODES ON DRIVEN GEAR HUB
F,ALL,FY,200                        * PUT FORCE IN THETA DIRECTION (CCW) ON NODES
/COM
NALL                                * RESTORE ALL NODES
/PNUM,NODE,0                        * TURN OFF NODE NUMBER DISPLAY
/PBC,ALL,1                          * TURN ON DISPLAY OF ALL BOUNDARY CONDITIONS
EUSEL,ELEM,1                        * UNSELECT SUPERELEMENT, WHICH WON'T PLOT
EPLOT                               * PLOT GAP ELEMENTS AND FORCE VECTORS ON HUBS
EALL                                * RESTORE ALL ELEMENTS
/COM
/COM                                DEFINE ANALYSIS
KAN,0                               * STATIC ANALYSIS
ITER,-3,1                           * SET NUMBER OF ITERATIONS AND PRINTS
AFWRITE                             * WRITE ANALYSIS FILE, FILE27
FINISH                              * END USE PASS ANALYSIS DEFINITION
/COM
/INPUT,27                           * BEGIN SOLUTION, READ ANALYSIS FILE
FINISH                              * END SOLUTION
/COM
/EOF                                * END SUBSTRUCTURE USE PASS
```

6.6 Stress Pass gear

```
/COM   ------------------------STRESS PASS-----------------------------------
/COM
/COM                                GET FILES 2, 11 FROM GENERATION (NOT USE) PASS
/GET,2,GGENT2                       * ASSIGN FILE GGENT2 TO UNIT NUMBER 2
/GET,11,GGENT11                     * ASSIGN FILE GGENT11 TO UNIT NUMBER 11
/COM                                GET FILE 13 FROM USE (NOT GENERATION) PASS
/GET,13,GUSET13                     * ASSIGN FILE GUSET13 TO UNIT NUMBER 13
/COM
/GET,12,GSTRT12                     * ASSIGN FILE GSTRT12 TO UNIT NUMBER 12
/GET,8,GSTRT8                       * ASSIGN FILE GSTRT8 TO UNIT NUMBER 8
/COPY,13,8                          * COPY MASTER DISPLACEMENT FILE TO FILE8
/COM
/STRESS,1                           * BEGIN STRESS PASS FOR SUPERELEMENT 1
/TITLE, SUBSTRUCTURE STRESS PASS
/COM
/COM                                CALCULATE STRESSES IN SUBSTRUCTURE
PRDISP,-1                           * SUPPRESS NODAL DISPLACEMENT PRINTOUT
ITER,3,1,1                          * EXPAND ITERATION 3
```

```
NSTRES,1                          * EXPAND 1 STRESS SOLUTION THIS STEP
END                               * END STEP
FINISH                            * END SOLUTION FOR STRESSES IN SUBSTRUCTURE
/COM
/EOF                              * END SUBSTRUCTURE STRESS PASS
```

6.7 Postprocessing Gear

```
/COM   -------------------------POSTPROCESSING---------------------------------
/COM
/POST1                            * BEGIN POSTPROCESSING
/TITLE, MODEL POSTPROCESSING
/SHOW,4336M                       * SET DISPLAY TYPE TO TEKTRONIX 4336M
/GET,12,GSTRT12                   * ASSIGN FILE GSTRT12 TO UNIT NUMBER 12
/COM
/COM                              SELECT ANALYSIS RESULTS
SET,1                             * USE LOAD STEP 1, LAST ITERATION
/COM
/COM                              SELECT ELEMENTS IN CONTACT AREA
CSYS,1                            * CYLINDRICAL COORDINATE SYSTEM
RIN = (D2/2)*0.8                  * DEFINE INNER RADIUS FOR PLOTS
ROUT = ((D1+D2)/2)*0.9            * DEFINE OUTER RADIUS FOR PLOTS
NSEL,X,RIN,ROUT                   * SELECT NODES BY RADIUS
NPLOT                             * PLOT NODES
NRSEL,Y,(-ANG2*4),(ANG2*4)        * RESELECT NODES BY ANGLE
NPLOT                             * PLOT NODES
CSYS,31                           * CYLINDRICAL CS AT CENTER OF GEAR 1
RC1 = (D1/2)-((1.25*D1)/N1)       * RADIUS OF CENTRAL REGION OF GEAR 1
NRSEL,X,RC1*0.7,D1+D2             * RESELECT NODES BY RADIUS
NPLOT                             * PLOT NODES
LOCAL,32,1,(D1+D2)/2,,,180        * CYL CS AT CENTER OF GEAR 1, ROTATED 180 DEG
TH1 = (-ANG1*3)*1.01              * STARTING ANGLE
TH1 =  (ANG1*3)*1.01              * ENDING ANGLE
NRSEL,Y,TH1,TH2                   * RESELECT NODES BY ANGLE
NPLOT                             * PLOT NODES IN CONTACT AREA, PLUS SOME
NUSE,NODE,        14              * UNSELECT THIS NODE (CHOSEN BY PICK OPTION)
NUSE,NODE,        20
NUSE,NODE,        21
NUSE,NODE,        22
NUSE,NODE,        23
NUSE,NODE,        26
NUSE,NODE,        28
NUSE,NODE,        29
NUSE,NODE,        30
NUSE,NODE,        31
NUSE,NODE,        35
NUSE,NODE,        36
NUSE,NODE,        37
NUSE,NODE,        38
NUSE,NODE,      1748
NUSE,NODE,      1750
NUSE,NODE,      1767
```

```
NUSE,NODE,      1770
NUSE,NODE,      1774
NUSE,NODE,      1782
ENODE                          * SELECT ELEMENTS WITH A NODE IN SELECTED SUBSET
EPLOT                          * PLOT ELEMENTS IN CONTACT AREA
/COM
/COM                           PLOT STRESS CONTOURS IN CONTACT AREA
/SHOW,4225,,1                  * SET DISPLAY TYPE TO 4225 TO GET CONTOUR LINES
/EDGE,,1                       * MAKE EDGE PLOT, REMOVING ELEMENT LINES
/NUM,1                         * TURN OFF NUMBERS ON STRESS CONTOURS
PLNSTR,SIGE                    * PLOT EQUIVALENT STRESS CONTOURS
FINISH                         * END POSTPROCESSING
/COM
/EOF                           * END SUBSTRUCTURE POSTPROCESSING RUN
```

6.8 Optimisation of a Fillet Radius

```
/COM
/COM       OPTIMIZATION OF A FILLET RADIUS
/COM
/COM       A stepped shaft under torsion is being analyzed to find
/COM       the minimum fillet radius which can be used. The objective
/COM       function is to minimize the manufacturing cost associated
/COM       with the fillet radius while satisfying a stress criterian.
/COM       Added to the optimization loop is an adaptive mesh capability.
/COM       This will ensure that the optimized mesh generated on the
/COM       solid model will not have an error norm greater than 5 per cent.
/COM
/COM
/COM       Set up parameters used by the analysis.
/COM
/COM       Model parameters.
/COM
DISP=2.E-3    * Maximum displacement seen at the end of shaft.
DDDD=0.        * Work parameter used in determing displacements.
ITRN=0         * Iteration counter.
RINN=1.        * Inner radius of the shaft.
ROUT=4.        * Outer radius of the shaft.
RSMA=1         * Size of radius step.
LFIL=1.        * Fillet radius (Optimization Design Variable).
L1=2           * Height of larger shaft.
L2=5           * Height of smaller shaft.
L3=0.          * Work parameter used in modelling.
/COM
/COM       Adaptive meshing (A.M.) parameters
/COM
TARG=.01       * Target value for A.M. error norm.
KSTA=2.        * Starting element size.
KFAC=3.        * Ratio of largest to smallest element size allowed.
KDK1=KSTA      * Initialazation of keypoint 1 element size.
KDK2=KSTA      *       "          "    "     2    "      "
KDK3=KSTA      *       "          "    "     3    "      "
```

```
KDK5=KSTA       *       "       "   "   5   "       "
KDK6=KSTA       *       "       "   "   6   "       "
KDK7=KSTA       *       "       "   "   7   "       "
KDK8=KSTA       *       "       "   "   8   "       "
KDK9=KSTA       *       "       "   "   9   "       "
FRAC=.9         * A.M. reduction factor of all keypoints.
FRA2=.5         * A.M. reduction factor of keypoint closest largest error norm.
KP=0            * Keypoint number closest largest error norm.
KSMA=0.         * Work parameter used to determine element size ratio.
ERR=1.0         * Error norm.
EEMX=0          * Work parameter used in error norm calculation.
ERMX=0          * Work parameter used in error norm calculation.
MFLG=0          * A.M. convergance flag.
/COM
/COM        Optimization parameters.
/COM
SMAX=0.         * Maximum Von Mises stress.
LFLL=0.         * Fillet radius parameter used as the objective function.
/COM
/COM        Top of loop. This call to Prep7 is needed because the optimization
/COM        copy would delete the :TOP label needed in the adaptive meshing
/COM        loop.
/COM
/PREP7
/OUT,26
RESUME,,,,1
FINI
/COM
/COM        While doing adaptive meshing, enter Prep7 and get keypoint closest
/COM        to element with largest error norm.
/COM
:TOP
*IF,ITRN,EQ,0,:STA
/PREP7
/OUT,26
RESUME,,,,1
*GET,KP,KPNE,EEMX
FINI
:STA
/COM
/COM        Enter Prep7 and build model.
/COM
/PREP7
/OUT,26
ET,1,83
MP,EX,1,3E7
MP,NUXY,1,.3
K,,RINN
K,,ROUT
K,,ROUT,L1
K,,RINN+RSMA,L1
K,,RINN+RSMA,L2
K,,RINN,L2
```

```
L3=LFIL/2.
L3=L3+L1
K,,RINN,L3
L,      1,      2
L,      2,      3
L,      3,      4
L,      4,      5
L,      5,      6
L,      6,      7
L,      7,      1
LFILLT,3,4,LFIL
AL,ALL
/COM
/COM      Begin adaptive meshing.
/COM
ITRN=ITRN+1
/COM
/COM      If the error norm is already less than the target value or this
/COM      is the first iteration skip the adaptive meshing.
/COM
*IF,MFLG,EQ,1,:SET
*IF,ITRN,EQ,1,:SET
/COM
/COM      Reduce all keypoint element sizes by the parameter FRAC.
/COM
KDK1=KDK1*FRAC
KDK2=KDK2*FRAC
KDK3=KDK3*FRAC
KDK5=KDK5*FRAC
KDK6=KDK6*FRAC
KDK7=KDK7*FRAC
KDK8=KDK8*FRAC
KDK9=KDK9*FRAC
/COM
/COM      Find the keypoint closest to the element with the largest error
/COM      norm and reduce its element size by FRA2. Also set the parameter
/COM      KSMA to be KFAC times the element size at that keypoint.
/COM
KSMA=KSTA
*IF,KP,EQ,0,:SET
*IF,KP,GT,1,:TWO
KDK1=KDK1*FRA2
KSMA=KFAC*KDK1
*GO,:SE2
:TWO
*IF,KP,GT,2,:THR
KDK2=KDK2*FRA2
KSMA=KFAC*KDK2
*GO,:SE2
:THR
*IF,KP,GT,3,:FOU
KDK3=KDK3*FRA2
KSMA=KFAC*KDK3
```

```
*GO,:SE2
:FOU
*IF,KP,GT,5,:FIV
KDK5=KDK5*FRA2
KSMA=KFAC*KDK5
*GO,:SE2
:FIV
*IF,KP,GT,6,:SIX
KDK6=KDK6*FRA2
KSMA=KFAC*KDK6
*GO,:SE2
:SIX
*IF,KP,GT,7,:SEV
KDK7=KDK7*FRA2
KSMA=KFAC*KDK7
*GO,:SE2
:SEV
*IF,KP,GT,8,:EIG
KDK8=KDK8*FRA2
KSMA=KFAC*KDK8
*GO,:SE2
:EIG
KDK9=KDK9*FRA2
KSMA=KFAC*KDK9
/COM
/COM      Now scan through all keypoint element size parameters and ensure
/COM      that none are greater than KSMA. This will help make smooth element
/COM      size transitions for meshing.
/COM
:SE2
*IF,KDK1,LE,KSMA,:AAA
KDK1=KSMA
:AAA
*IF,KDK2,LE,KSMA,:BBB
KDK2=KSMA
:BBB
*IF,KDK3,LE,KSMA,:CCC
KDK3=KSMA
:CCC
*IF,KDK5,LE,KSMA,:DDD
KDK5=KSMA
:DDD
*IF,KDK6,LE,KSMA,:EEE
KDK6=KSMA
:EEE
*IF,KDK7,LE,KSMA,:FFF
KDK7=KSMA
:FFF
*IF,KDK8,LE,KSMA,:GGG
KDK8=KSMA
:GGG
*IF,KDK9,LE,KSMA,:SET
KDK9=KSMA
```

```
/COM
/COM      Set keypoint element sizes with the use of the KDVS command.
/COM
:SET
KDVS,1,KDK1
KDVS,2,KDK2
KDVS,3,KDK3
KDVS,5,KDK5
KDVS,6,KDK6
KDVS,7,KDK7
KDVS,8,KDK8
KDVS,9,KDK9
/COM
/COM     Set line divisions.
/COM
LDVA,,1.5,5
LDVS,ALL,30,ANGLE,,2
/COM
/COM     Mesh and apply displacements.
/COM
AMESH,1
MODE,0,1
KD,1,UZ,,,1
KD,2,UZ,,,1
KD,1,UY,,,1
KD,2,UY,,,1
DDDD=RSMA/RINN
DDDD=DDDD+1.
KD,5,UZ,DISP*DDDD,,1
KD,6,UZ,DISP,,1
/COM
/COM    Set strain energy flag and write solution input deck.
/COM
KSE,1
ITER,1,0
AFWRITE
FINISH
/COM
/COM    Solve model.
/COM
/INP,27
FINI
/POST1
/OUT,26
STRESS,ENRM     *Store element error norms.
STRESS,SENE     *Store Von Mises stress.
SET             *Store load step results.
PRNST
SSUM
LFLL=LFIL
/COM
/COM    Calculate error norms.
/COM
```

```
*GET,ENR,SSUM,ENRM
*GET,TSE,SSUM,SENE
ERR=(ENR/(TSE+ENR))
ERR=SQRT(ERR)
/COM
/COM    Find element with largest error norm.
/COM
ESORT,ENRM,,1,1
*GET,ERMX,MAX
ESEL,ENRM,ERMX
*GET,EEMX,ELMX
EALL
NSEL,SIGE,1,1.E8
NSORT,SIGE
*GET,SMAX,MAX    *Get maximum Von Mises Stress.
NALL
FINI
/COM
/COM   If the error norm is greater than the target, go back and remesh
/COM   model with a finer mesh
/COM
*IF,ERR,GT,TARG,:SFLG
MFLG=1.
:SFLG
*IF,MFLG,EQ,0,:TOP
/OPT
/OUT
/NOPR
OPVAR,LFIL,DV,.1,1.5
OPVAR,SMAX,SV,0,30.E3
OPVAR,LFLL,OBJ
/GOPR
OPLIST,,,1
OPCOPY
OPRUN,30
OPLIST,ALL,,1
FINI
```

6.9 Plastic Blower Wheel Housing

```
/PREP7
/TITLE, BLOWER HOUSING
/COM        DEFINE SIDE PROFILE OF HOUSING
K,100
K,101,,,1
K,102, 1,2
CIRCLE,100,10, 101,102
K,101,1,-1
CIRCLE,101,12.5
K,9, -20,18
K,10, -20,6
L,9,10
```

```
K,11, -5.5,18
LARC,5,11,3, 19
LDEL,5
L,9,11
KGEN,2, 10,,, 13
L,10,12
LINT,11,6
LDEL,6,12,6, 1
LFIL,11,13, 1
LDIV,10
L,2,12
L,1,14
AL,1,14,6,11,9,5,12,15
AL,2,3,4,15,10,8,7,13,14
/COM        CREATE NODES AND ELEMENTS ON PROFILE
LDVS,ALL,40,ANGLE
LDVS,ALL,4,,,2
LDVA,,2,2
ET,1,93
AMESH,1,2
/COM        DUPLICATE AREAS AND FE MESH TO OPPOSITE SIDE
AGEN,2, 1,2,, 0,0,12
/COM        CREATE REMAINING AREAS
L,9,20
ADRAG,31,,,,,, 5,12,10,8,7,13
ADRAG,49,,,,,, 6,11
NUMMRG,KPOI
/COM        MESH REMAINING AREAS
LDVS,ALL,40,ANGLE
LDVS,ALL,4,,,2
LDVA
AMESH,5,12
FINISH
```

6.10 Retaining Ring Model

```
/prep7

/title, Retaining ring
orad = 22                       * Outer radius
irad = 18                       * Inner radius
hrad = 2                        * Hole radius
/show,snapr,f33,1
/ploff,0,0,1,0
/num,2
kplot

k,1,,orad                       * System 11 (cylindrical) for outer surface
k,2,,orad*2
k,3,-orad,orad
cskp,11,1,1,2,3
csys,0
```

```
k,5,,((orad+irad)/2)            * System 12 (cylindrical) for inner surface
cskp,12,1,5,2,3
/psym,csys,1
/pnum,kpoi,1
kplot

csys,11
k,3,orad,165                    * Points on outer surface
kfill,2,3,1,4
kplot

1,2,4                           * Outer surface line segments
1,4,3
csys,12
k,6,irad                        * Inner surface keypoints
k,7,irad,130
kfill,6,7,1,8
1,6,8                           * Inner surface line segments
1,8,7
csys,0
k,,-irad/2,7                    * Keypoint at center of hole
circle,9,hrad                   * Hole surface
circle,9,5                      * Small circular part of inner surface
lang,12,5                       * Tangent from center of inner surface
/com                              system to small circle
csys,12
1,3,18
ldel,14
LDEL,    12,    12,1,    0
LDEL,    11,    11,1,    0
lcom,9,13
linter,4,10                     * Fillet between small and large radii
lfil,4,10,4
1,2,6
LDEL,    12,    12,1,    0
LDEL,    11,    11,1,    0
kpls                            * Delete unused keypoints
kpinv
kdel,all
numcmp,all                      * Compress out unused keypoint, line numbers
kpall
csys,0
L,    10,     7                 * Divide region to allow AL areas
L,     6,    11
al,5,14,9,15                    * Define area 1
LSUS,LINE,         9            * Select line set for area 2
LSUS,LINE,         5
al,all                          * Define area 2
arse,,1,2,,1                    * Select all active areas, lines, points
/pnum,line,1
/pnum,area,1
aplot
```

```
nummrg,kpoi                    * Note: no effect on this model.

numcmp,all                     * Note: no effect on this model.

et,1,2                         * 6-node triangular stress solid
aplot

ldvs,all,15,angle              * 15 degrees/element maximum
kdvs,2,1                       * Element size near keypoint 2 = 1
ldva                           * Adjust element divisions for smooth
/com                             transitions
lplot

amesh,all                      * Create nodes and elements in areas 1-2
eplot

waves                          * Reordering without a starting list

csys,11                        * Symmetry constraints on symmetry plane
kpse,y
lskp,1
lsbc,all
kprs,x,orad                    * Complete restraint at one point
kd,all,all
kpse,x,orad                    * Pressure on outer surface
lskp,1
lpsf,all,100
arse,,1,2,,1                   * Select entire solid model and finite
/com                             element model
sbctran                        * Transfer boundary conditions to finite
/com                             element model for plotting
/pbc,all,1
nplot
finish
```

6.11 20 – Node Bricks Junction with 10 – Node Tetrahedra (Top Model)

```
/prep7
/title, brick and tetrahedron junction- stif95/92
k,1
k,2,1
k,3,1,1
k,4,,1
kgen,3,all,,,,,1
a,1,2,3,4
a,9,10,11,12
v,1,2,3,4,5,6,7,8
v,5,6,7,8,9,10,11,12
et,1,95
ldvs,all,,4
vmesh,1
et,2,92
```

```
type,2
vmesh,2
waves
ex,1,3e7
asbc,1
kd,1,all
kd,2,uy
apsf,2,-1000
afwrite
finish
/input,27
finish
/post1
set,1,1
/show,bktet1,f33,1
/type,,2
/view,,2,3,1
plns,sige
finish
```

6.12 8 – Node Brick Junction with 20 – Node Tetraheda (Bot Model)

```
/prep7
/title, brick and tetrahedron junction- stif45/92
k,1
k,2,1
k,3,1,1
k,4,,1
kgen,3,all,,,,,1
a,1,2,3,4
a,9,10,11,12
v,1,2,3,4,5,6,7,8
v,5,6,7,8,9,10,11,12
et,1,45
ldvs,all,,4
vmesh,1
et,2,92
type,2
vmesh,2
waves
ex,1,3e7
asbc,1
kd,1,all
kd,2,uy
apsf,2,-1000
afwrite
finish
/input,27
finish
/post1
set,1,1
/show,bktet2,f33,1
```

```
/type,,2
/view,,2,3,1
plns,sige
finish
```

6.13 Volume Generation with Continuous Mesh

```
/prep7
/title, VOLUME GENERATION WITH CONTINUOUS MESH
csys,1
k,1,10,90                          * Build volume 1.
k,2,10,90,2
k,3,11.5,90
kgen,2,3,,,,,1
kgen,2,4,,,-1
kgen,2,2,,,.5
L,      6,      5
L,      5,      4
L,      4,      3
L,      3,      1
L,      1,      2
L,      2,      6
lfil,1,2,.3
lfil,4,5,.6
k,101
k,102,,,1
al,all
arot,1,7,2,3,4,8, 101,102,7.5
arot,5,6,,,,, 101,102,7.5
nummrg,kpoi
kpse,y,97.5
lskp,1
al,all
arse,,1,99,,1
kpinv
kdel,all
arse,,1,99,,1
va,all                             * Volume 1 complete.
ldvs,all,.3                        * Establish element density.
/com
et,20,2                            * Mesh area 1 with 6-noded triangles.
type,20
amesh,1
agen,2, 1,,, 0,7.5                 * Copy area 1 and triangle mesh to
/com                                 opposite side.
nummrg,kpoi                        * Merge duplicated area into the volume.
/view,,1,2,3
/ploff,1,1,1,1,1
/num,2
/pnum,area,1
/noer
/user
```

```
et,1,92                          * Mesh volume 1 with tetrahedra.
type,1
vmesh,1
arse,type,20                     * Clear area elements.
aclear,all
arall
vgen,3, 1,,, 0,7.5               * Generate the remaining volumes, nodes,
/com                                and element.
nummrg,all                       * Merge the segments together.
numcmp,all                 * Compress the data.
/erase                           * Plot the final model.
/auto
/type,,2
/show,volgen,f33,1
eplot
finish
```

6.14 2 – Dimensional Contact Example

```
/prep7
/title, 2-DIMENSIONAL CONTACT EXAMPLE
/com
k,1,,1                     * Create the solid model.  Make sure that keypoints
k,2,5,1                    *  and line segments line up across proposed contact
k,3,0,0                    *  surface, but are not coincident.
k,4,2
k,5,3
k,6,5
L,      1,      3
L,      3,      4
L,      1,      2
L,      2,      6
L,      6,      5
kgen,2, 4,5,, 0,.2
L,      4,      7
L,      7,      8
L,      8,      5
al,all
kpse,y
kgen,2, all,,, 0,-.3
k,13,6,-.3
kgen,2, 9,13,4, 0,-.6
L,     14,      9
L,      9,     10
L,     10,     11
L,     11,     12
L,     12,     13
L,     13,     15
L,     15,     14
lskp,1
lsus,,2,5
al,all
```

```
arse,,1,2,, 1                    * Solid model is complete.
/com
ldvs,all,.4,,,2                  * Establish element divisions.
ldva
/show,linegap,f33,1              * Plot the initial solid model.
/ploff,1,1,1,1,1
/pnum,line,1
lplot

LSSE,LINE,          2            * Select line segments on one contact surface
LSAS,LINE,          5              (lsse,pick recommended).
/com
/com                             Mesh contact surface 1 with line elements.
/com                              Establish n1s1 and n2s1 as starting and
/com                              ending node numbers on surface 1.
/com
et,20,1                          * Spar element
type,20
n1s1 = 1
lmesh,all
*get,n2s1,nmax
/com                             Duplicate contact surface 1 onto contact
/com                              surface 2.
n1s2 = n2s1 + 1
lgen,2, all,,, 0,-0.3            * Geometric increment must put new lines
/com                               on top of existing surface 2 lines.
*get,n2s2,nmax
/com                             Merge new lines into surface 2.
/com
kpall                            * Select all keypoints for merge.
nummrg,kpoi                      * Merge selected keypoints.
ldva                             * Adjust surrounding lines again.
/com
/com                             Mesh the rest of the model.
/com
et,1,82                          * 8-noded quadrilateral stress solid
type,1
arse,,1,2,, 1
amesh,all
/shri,.3                         * Plot the area and line elements.
eplot
/noer
/user
/pnum,node,1
esel,type,20
eplot
eall
/auto
/erase

/com                             Connect gap elements between the surfaces.
/com
et,2,12                          * 2-D interface element
```

```
type,2
e,n1s1,n1s2
egen,((n2s1-n1s1)+1),1,-1
et,20,0                          * Deactivate spar elements
numcmp,all                       * Compress the data bases.
/pnum,node                       * Plot the final finite element model.
eplot
finish
```

6.15 3 – Dimensional Contact Example

```
/prep7
/title, 3-DIMENSIONAL CONTACT EXAMPLE
/com
k,1,,1                    * Create the solid model.  Make sure that keypoints,
k,2,5,1                   *  line segments, and areas line up across proposed
k,3,0,0                   *  contact surface, but that surfaces are separated
k,4,2                     *  enough that keypoints will not merge during
k,5,3                     *  NUMMRG,KPOI.
k,6,5
L,      1,      3
L,      3,      4
L,      1,      2
L,      2,      6
L,      6,      5
kgen,2, 4,5,, 0,.2
L,      4,      7
L,      7,      8
L,      8,      5
al,all
kpse,y
kgen,2, all,,, 0,-.3
k,13,6,-.3
kgen,2, 9,13,4, 0,-.6
L,     14,      9
L,      9,     10
L,     10,     11
L,     11,     12
L,     12,     13
L,     13,     15
L,     15,     14
lskp,1
lsus,,2,5
al,all
arse,,1,,,1
k,101
k,102,,,.6
l,101,102
adrag,1,2,6,7,8,5, 16
adrag,4,3,,,,,     16
lsus,,16
kpls
```

```
nummrg,kpoi
kpse,z,.6
lskp,1
al,all
va,all
arse,,2,,,1
lsas,,16
adrag,9,10,11,12,13,14, 16
adrag,15,,,,,,          16
nummrg,kpoi
kpse,z,.6
kprs,y,-99,-.1
lskp,1
al,all
va,all
vlse,,1,2,, 1                   * Solid model is complete.
ldvs,all,.8,,,2                 * Establish element division.
ldva
/show,areagap,f33,1             * Plot the initial solid model.
/ploff,1,1,1,1,1
/pnum,area,1
/view,,1,2,4
/type,,2
aplot
ARSE,,   4                      * Select areas on one contact surface
ARAS,,   8                      *  (arse,pick recommended).
/com
/com                            Mesh contact surface 1 with area elements.
/com                             Establish n1s1 and n2s1 as starting and
/com                             ending node numbers on surface 1.
/com
et,20,63                        * 4-noded shell element
type,20
n1s1 = 1
amesh,all
*get,n2s1,nmax
/com                            Duplicate contact surface 1 onto contact
/com                             surface 2.
n1s2 = n2s1 + 1
agen,2, all,,, 0,-0.3           * Geometric increment must put new areas
/com                               on top of existing surface 2 areas.
*get,n2s2,nmax
/com                            Merge new areas into surface 2.
/com
kpall                           * Select all keypoints for merge.
nummrg,kpoi                     * Merge selected keypoints.
ldva                            * Adjust surrounding lines again.
/com
/com                            Mesh the rest of the model.
/com
et,1,92                         * 10-noded tetrahedron stress solid
type,1
vlse,,1,2,, 1
```

```
vmesh,all
/shri,.3                          * Plot the volume and area elements.
eplot
/noer
/user
/pnum,node,1
/type
esel,type,20
eplot
eall
/auto
/erase
/type,,2
/com                              Connect gap elements between the surfaces.
/com
et,2,12                           * 2-D interface element
type,2
e,n1s1,n1s2
egen,((n2s1-n1s1)+1),1,-1
et,20,0                           * Deactivate spar elements
numcmp,all                        * Compress the data bases.
/com
/pnum,node                        * Plot the final finite element model.
eplot
/noer
/user
/type
esel,type,2
eplot
eall
/auto
/erase
finish
```

6.16 Connecting Dissimilar Meshes with Constraint Equation Interface

```
/prep7
/title, Dissimilar shapes with constraint equation interface
k,1
k,2,.7
k,3,.7,.3
k,4,,.3
kgen,2,all,,,,,1
v,1,2,3,4,5,6,7,8
k,,.2,.3,.8
kgen,2,9,,, .3
1,9,10
kgen,2,10,,, .2
kgen,2,11,,, ,.1
lgen,2,13,,, ,.3
A,     9,    13,    14,    10
vrot,7,,,,,, 11,12,-70
```

```
et,1,95
elsi,.3,,2
vmesh,1
elsi,.1,,2
vmesh,2
vlse,,2,,,1
nrse,y,.3
vlse,,1
evol
ceint
vlse,,1,2,,1
ex,1,3e7
asbc,2
aabc,2
apsf,9,-1000
afwrite
/ploff,1,,1
/num,2
/show,cein1,f33,1
lplot
/noer
/user
/shri,.3
/type,,2
eplot
/erase
/auto
finish
/input,27
finish
```

6.17 Primitive Geometry USER – FILE for Gear Tooth Profile

```
:COM   THIS DATA BLOCK IS USED TO DEFINE
:COM   A COARSE PITCH INVOLUTE, 20 DEG GEAR TOOTH PROFILE PRIMITIVE
:COM
:COM   TO ENTER A PRIMITIVE:
:COM    *USE,GTOOTH,D,N
:COM
:COM   INPUT DESCRIPTIONS:
:COM    D      - PITCH DIAMETER (IN.) (ARG1)
:COM    N      - NUMBER OF TEETH (ARG2)
:COM
:COM   ASSUMPTIONS:
:COM   -DATA BLOCK IS FOR USE IN THE ANSYS PROGRAM, REVISION 4.4
:COM   -SOLID MODEL COMPONENTS WILL BE DEFINED AT END OF PRESENT GEOMETRY
:COM   -ALL NUMSTR VALUES WILL BE RESET TO DEFAULTS
:COM   -CURRENT LOCAL COORDINATE SYSTEM (LCS) IS DEFINED BEFORE USING USER FILE
:COM   -LCS IS A CYLINDRICAL TYPE WITH PARM=1.0
:COM   -GEAR HUB IS CENTERED ON ORIGIN OF LCS
:COM   -GEAR TOOTH IS CENTERED ON THETA=0 IN LCS
:COM   -LOCAL COORDINATE SYSTEM 40 WILL BE USED AND THEN DELETED
```

```
:COM   -LCS WILL BE RESET
:COM   -NUMBER OF TEETH IS LESS THAN OR EQUAL TO 41
:COM   -N/D IS LESS THAN 20
:COM   -THIS FILE WILL CREATE PARAMETERS
:COM     RDBC = RADIUS OF BASE CIRCLE (IN.)
:COM     RDAC = RADIUS OF ADDENDUM CIRCLE (IN.)
:COM     RDRC = RADIUS OF ROOT CIRCLE (IN.)
:COM     TANG = ANGLE SUBTENDED BY TOOTH = 1/2 PITCH ANGLE (DEG)
:COM   -GEAR TOOTH NOMENCLATURE FOLLOWS
:COM     MARKS' STANDARD HANDBOOK FOR MECH. ENGS., MCGRAW-HILL, 9TH ED., AND
:COM     SHIGLEY, J. E., MACHINE DESIGN, MCGRAW-HILL, 1956
:COM
/COM
/COM  DEFINING A GEAR TOOTH PROFILE PRIMITIVE
/COM
/NOPR                           * TURN OFF PRINT OF INTERPRETED INPUT DATA
AR10 = 0 $AR11 = 0              * INITIALIZE MAX KEYPOINT AND MAX LINE SEGMENT
*GET,AR10,MXKP                  * GET MAX KEYPOINT NUMBER
*GET,AR11,MXLS                  * GET MAX LINE SEGMENT NUMBER
*GET,AR12,CSYS                  * GET CURRENT COORDINATE SYSTEM NUMBER
NUMSTR,KPOI,AR10+1              * START NEW KEYPTS AFTER EXISTING KEYPTS
NUMSTR,LINE,AR11+11             * START LINES AT EXISTING MAX PLUS 11
:COM
AR13 = 20                       * PRESSURE ANGLE (DEG)
AR14 = ARG2/ARG1                * DIAMETRAL PITCH
AR15 = (1)/AR14                 * ADDENDUM
AR16 = ACOS(-1)/180             * DEG TO RAD CONVERSION, PI/180
AR17 = (1.25)/AR14              * DEDENDUM
AR18 = (0.3)/AR14               * FILLET RADIUS
RDBC = (ARG1/2)*(COS(AR13*AR16))  * RADIUS OF BASE CIRCLE
RDAC = (ARG1/2)+AR15            * RADIUS OF ADDENDUM CIRCLE
RDRC = (ARG1/2)-AR17            * RADIUS OF ROOT CIRCLE
TANG = (180)/ARG2               * TOOTH ANGLE (DEG)
:COM
:COM                            APPROXIMATE INVOLUTE THROUGH KEYPOINTS A, B, C
K,,RDBC,0                       * KEYPOINT A
AR19 = (ARG1/2)**2              * SQUARE OF RADIUS OF PITCH CIRCLE
AR20 = RDBC**2                  * SQUARE OF RADIUS OF BASE CIRCLE
AR21 = SQRT(AR19-AR20)/RDBC     * GAMMA B (RAD)
AR22 = AR21-ATAN(AR21)          * THETA B (RAD)
K,,(ARG1/2),AR22/AR16           * KEYPOINT B
AR23 = RDAC*1.01                * RADIUS OF C, SLIGHTLY GT RADIUS OF ADD CIRCLE
AR19 = AR23**2                  * SQUARE OF RADIUS OF C
AR24 = SQRT(AR19-AR20)/RDBC     * GAMMA C (RAD)
AR25 = AR24-ATAN(AR24)          * THETA C (RAD)
K,,AR23,AR25/AR16               * KEYPOINT C
LOCAL,40,0                      * CARTESIAN COORDINATE SYSTEM
AR26 = TAN(AR24)                * SLOPE AT C
SPLINE,AR10+3,AR10+2,AR10+1,,,,1,AR26,0,0,0,0  * CONNECT A, B, C WITH 2 SPLINES
LCOMB,AR11+11,AR11+12           * COMBINE 2 SPLINES INTO 1 LINE
:COM
:COM                            CONSTRUCT TOP LAND THROUGH KEYPOINTS H AND J
CSYS,AR12                       * CYLINDRICAL COORDINATE SYSTEM
```

```
K,,RDAC,(AR22/AR16)+(TANG/2)     * KEYPOINT H AT CENTER OF TOP LAND
K,,RDAC,AR22/AR16                * KEYPOINT I ON TOP LAND EXTENSION
L,AR10+2,AR10+4                  * TOP LAND ARC
LINT,AR11+11,AR11+12             * INTERSECT INVOLUTE AND TOP LAND ARC AT J
LDEL,AR11+11,AR11+14,3           * DELETE EXTRA LINES FORMED BY INTERSECTION
:COM
:COM                             CONSTRUCT RADIAL EXTENSION OF INVOLUTE FROM BASE
:COM                              CIRCLE AT KEYPT A TO ROOT CIRCLE AT KEYPT D
K,,RDRC,0                        * KEYPT D AT BOTTOM LAND - FLANK INTERSECTION
L,AR10+1,AR10+6                  * RADIAL EXTENSION OF FLANK
LCOM,AR11+13,AR11+11             * COMBINE INVOLUTE AND RADIAL EXTENSION
:COM
:COM                             CONSTRUCT BOTTOM LAND THROUGH KEYPOINTS D AND E
K,,RDRC,(AR22/AR16)-(TANG/2)     * KEYPOINT E AT CENTER OF BOTTOM LAND
L,AR10+6,AR10+1                  * BOTTOM LAND ARC
:COM
:COM                             CONSTRUCT FILLET AT BOTTOM LAND - FLANK
LFIL,AR11+13,AR11+11,AR18        * FILLET CREATING KEYPTS G (FLANK), F (BOT LAND)
KPLS $KPINV                      * SELECT UNUSED KEYPOINTS
KDEL,AR10+1,AR10+99              * DELETE UNUSED KEYPOINTS
:COM
:COM                             ROTATE INTO POSITION
NUMSTR,LINE,AR11+1               * START LINES AT USER'S MAX PLUS 1
AR27 = -((AR22/AR16)+(TANG/2))   * ROTATION (DEG) TO CENTER TOOTH ON THETA = 0
LGEN,2,AR11+11,AR11+14,1,,AR27   * ROTATE ALL LINES
LDEL,AR11+11,AR11+14,,1          * DELETE ORIGINAL SET OF LINES
:COM
:COM                             RENUMBER KEYPOINTS CONSECUTIVELY
KGEN,2,AR10+3                    * GENERATE KEYPOINT AT F
KGEN,2,AR10+9                    * GENERATE KEYPOINT AT J
LMOD,AR11+4,,AR10+1              * MODIFY BOTH KEYPOINT NUMBERS OF LINE
LMOD,AR11+1,AR10+1               * MODIFY FIRST KEYPOINT NUMBER OF LINE
KDEL,AR10+3                      * DELETE UNUSED KEYPOINT
KGEN,2,AR10+4                    * GENERATE KEYPOINT AT E
LMOD,AR11+1,,AR10+3              * MODIFY LAST KEYPOINT NUMBER OF LINE
KDEL,AR10+4                      * DELETE UNUSED KEYPOINT NUMBER
KGEN,2,AR10+10                   * GENERATE KEYPOINT AT G
LMOD,AR11+3,AR10+2,AR10+4        * MODIFY BOTH KEYPOINT NUMBERS OF LINE
LMOD,AR11+4,AR10+4               * MODIFY FIRST KEYPOINT NUMBER OF LINE
KGEN,2,AR10+6                    * GENERATE KEYPOINT AT H
LMOD,AR11+2,AR10+5,AR10+2        * MODIFY BOTH KEYPOINT NUMBERS OF LINE
KDEL,AR10+6,AR10+10              * DELETE UNUSED KEYPOINTS
:COM
KPALL $LSALL                     * RESTORE ALL KEYPOINTS AND LINES
CSDEL,40                         * DELETE TEMPORARY COORDINATE SYSTEM
CSYS,AR12                        * ACTIVATE USER'S COORDINATE SYSTEM
NUMSTR,DEFA                      * RESTORE DEFAULT NUMBERING SCHEME
/COM
/COM  PROFILE COMPLETE
/GOPR                            * TURN ON PRINT OF INTERPRETED INPUT DATA
```

7. Composite Structures

7.1 Clamped Plate Analysis with Unsymmetriccally Stacked Laminate

```
C*************************************************************************
C*** EXAMPLE 1 : CLAMPED PLATE ANALYSIS WITH UNSYMMETRIC STACKING  ***
C***              SEQUENCE (0/45/-45/90) USING STIF99 ELEMENTS      ***
C*************************************************************************
C***.....................................................................***
C***            P R E P R O C E S S I N G   P H A S E                  ***
/PREP7
C***=====================================================================
/TITLE, CLAMPED LAMINATE UNDER UNIFORM PRESSURE AND THERMAL LOAD
C***=====================================================================
C***.....................................................................***
C*** DEFINE ELEMENT TYPE, OUTPUT CONTROLS AND LAMINATE DATA         ***
ET,1,99,,5,,,1                  *PRINT MATERIAL PROPERTY MATRICES
ET,2,99,,,,,2,4                 *DEFINE TYPE 2 FOR PRINTOUT CONTROL
R,1,4                           *PRINT STRESS AND STRAIN LAYER
RMORE                           *SOLUTION AT CORNER NODES FOR TYPE 2
RMORE,1,  0,0.1, 1,45,0.1       *(0/45/-45/90) STACKING WITH UNIFORM
RMORE,1,-45,0.1, 1,90,0.1       *THICKNESS AND IDENTICAL MATERIAL
C***.....................................................................***
C*** DEFINE MATERIAL PROPERTIES                                      ***
MP,EX,1,1.5E6
MP,EY,1,0.3E5                   *
MP,NUXY,1,0.006                 *ORTHOTROPIC MATERIAL PROPERTIES
MP,GXY,1,0.2E5                  *
MP,ALPX,1,2E-4
MP,ALPY,1,1E-4
C***.....................................................................***
C*** DEFINE NODES AND ELEMENTS USING QUADRILATERAL MESHER            ***
K,1                             *10X10 SQUARE PLATE; 0.4 THICK
K,2,10
K,3,10,10
K,4,,10
A,1,2,3,4
LOCAL,11                        *DEFINE LOCAL C.S. = GLOBAL C.S.
ESYS,11                         *ASSIGN C.S. 11 TO ESYS ATTRIBUTE
ELSIZE,,8                       *8 X 8 MESH USING 8 NODE SHELL STIF99
AMESH,1                         *NOTE - ALL ELEMENTS GENERATED WILL
NSEL,X,3.74,5.1
NRSEL,Y,3.74,5.1                *SELECT NODES NEAR PLATE CENTER
ENODE,1                         *SELECT ELEMENT CONTAINING CENTER NODE
TYPE,2
EMODIF,ALL                      *MODIFY THIS ELEMENT TO TYPE 2
NALL                            *FOR SELECTIVE SOLUTION PRINTOUT
EALL                            *RESTORE ALL NODES AND ELEMENTS
C***.....................................................................***
C*** APPLY BOUNDARY CONDITONS, PRESSURE AND THERMAL LOAD             ***
NSEL,X                          *HAVE ELEMENT C.S. PARALLEL TO GLOBAL
```

```
NASEL,X,10
NASEL,Y                          *SELECT NODES ALONG THE PLATE BOUNDARY
NASEL,Y,10
D,ALL,ALL                        *CLAMP ALL SELECTED NODES
NALL
EP,ALL,2,10                      *APPLY UNIFORM ELEMENT PRESSURE
TE,ALL,30,60                     *APPLY BOTTOM AND TOP SURFACE TEMPS.
TREF,20                          *REFERENCE TEMPERATURE
KTEMP,-1                         *USE SPECIFIED ELEMENT TEMPERATURE
C***.....................................................***
C*** PRINTOUT AND POSTDATA CONTROL                       ***
ITER,1,1,1
POST,1,1,4,2,4                   *STORE LAYER DATA (LEVEL 4) IN FILE12
PRSTR,-1,1                       *NO SOLUTION PRINTOUT FOR TYPE 1
PRSTR,,2                         *PRINT SOLUTION ONLY FOR ELEM TYPE 2
PRDISP,-1                        *SUPPRESS DISPLACEMENT PRINTOUT
PRRF,-1                          *SUPPRESS REACTION FORCE PRINTOUT
C***.....................................................***
C*** CHECK DATA AND WRITE ANALYSIS FILE                  ***
AFWRIT
FINI
C***.....................................................***
C***              S O L U T I O N   P H A S E            ***
/INP,27
FINI
C***.....................................................***
C***         P O S T - P R O C E S S I N G   P H A S E   ***
/POST1
SET,1,1
NSEL,Y,5                         *SELECT NODES ALONG MID-SPAN
PRDISP                           *PRINT DISPLACEMENTS
TOP                              *SELECT ELEMENT TOP FACE
PRNST,COMP                       *PRINT STRESSES ALONG MID-SPAN
BOT                              *SELECT ELEMENT BOTTOM FACE
PRNSTR,COMP
NSEL,X                           *SELECT EDGE ALONG X-AXIS
PRRF                             *PRINT REACTION FORCES
TOP                              *SELECT ELEMENT TOP FACE
PRNST,COMP                       *PRINT STRESSES ALONG MID-SPAN
BOT                              *SELECT ELEMENT BOTTOM FACE
PRNSTR,COMP
PRPATH,1,2,SXY                   *VARIATION OF SXY ALONG X-AXIS EDGE
NALL
C***.....................................................***
C*** PLOT DISPLACEMENT AND STRESS FIELDS
/SHOW,FILE33,,1                  *WRITE PLOTS TO FILE33
/DSCA,1,100                      *SCALE TO 100 TIMES
/TITLE, IN-PLANE DEFORMATIONS DUE TO COUPLING EFFECTS
PLDISP                           *PLOT IN-PLANE DISPLACEMENTS
TOP                              *SELECT TOP STRESSES
/TITLE, SX - STRESS FIELD AT TOP SURFACE IN GLOBAL C.S.
PLNS,SX
/TITLE, SY - STRESS FIELD AT TOP SURFACE IN GLOBAL C.S.
```

```
PLNS,SY
/TITLE, SXY - STRESS FIELD AT TOP SURFACE IN GLOBAL C.S.
PLNS,SXY
BOT                              *SELECT BOTTOM STRESSES
/TITLE, SX - STRESS FIELD AT BOTTOM SURFACE IN GLOBAL C.S.
PLNS,SX
/TITLE, SY - STRESS FIELD AT BOTTOM SURFACE IN GLOBAL C.S.
PLNS,SY
/TITLE, SXY - STRESS FIELD AT BOTTOM SURFACE IN GLOBAL C.S.
PLNS,SXY
/VIEW,1,1,1,1                    *CHANGE VIEW DIIRECTION
/VUP,1,Z
/DSC,1
/TITLE, DEFORMATIONS IN A (0/45/-45/90) CLAMPED PLATE
PLDISP
/TITLE, VARIATION OF SXY ALONG X-AXIS EDGE
/VIEW,1
/VUP
PLPATH,1,2,SXY                   *EDGE ELONG X-AXIS
C***.....................................................................***
C*** RESET AND PROCESS LAYER DATA
RESET
LAYER,3                          *SELECT LAYER 3 (-45 DEG LAYER)
SET,1,1
NSEL,Y,5                         *SELECT MID-SPAN OF PLATE
PRNST,COMP                       *PRINT LAYER MID STRESSES IN LAYER C.S
GLOBAL,0                         *CHOOSE LAYER CS FOR LAYER OUTPUT
PRNST,COMP                       *PRINT LAYER MID STRESS IN GLOBAL
CSYS,11                          *CHOOSE ELEMENT C.S (LOCAL 11=ESYS)
PRNST,COMP
NALL
/VIEW,1
CSYS,0
GLOBAL,1                         *USE GLOBAL COORDINATES
/TITLE, SX STRESSES AT MID OF LAYER 3 IN GLOBAL C.S
PLNS,SX
GLOBAL,0
/TITLE, SX STRESSES AT MID OF LAYER 3 IN LAYER C.S
PLNS,SX
FINI
C*************************************************************************
```

7.2 Interlaminar Shear Stress in Thick Cantilever Beam

```
C*************************************************************************
C*** EXAMPLE 2 : INTERLAMINAR SHEAR IN A THICK SANDWICH BEAM        ***
C***              COMPARISON OF STIF99 AND STIF46                   ***
C*************************************************************************
C***                                                                ***
/PREP7
C***=================================================================***
C*** M O D E L   1 : 6 STIF99 ELEMENTS ALONG BEAM LENGTH            ***
```

```
C***                  16 LAYERS PER ELEMENT                          ***
C***=========================================================================***
/NOPR
ET,1,99,,,,,1,,1                    *TYPE1 SUPPRESS PRINTOUT
ET,2,99,,,,,1,4                     *TYPE2 FOR PRINTOUT; STORE STRESSES
C***.....................................................................***
C*** REAL CONSTANTS - DEFINE SANDWICH STACKING SEQUENCE                 ***
R,1,16,1                            *16 SYMMETRIC LAYERS
RMORE                               *(45/45/-45/-45/0/0/0/0)SYMM
RMORE,1, 45,.125, 1, 45,.125        *EACH OF TK=0.125
RMORE,1,-45,.125, 1,-45,.125        *(ONLY 8 ARE SPECIFIED - LSYM=1)
RMORE,2,,.125,2,,.125
RP2
C***.....................................................................***
C*** DEFINE MATERIAL PROPERTIES FOR PLY AND CORE                        ***
MP,EX,1,30E6                        *MAT1: PLY MATERIAL PROPERTIES
MP,EY,1,12E6                        *      (ORTHOTROPIC)
MP,NUXY,1,0.12
MP,GXY,1,7E6
MP,EX,2,3E6                         *MAT2: CORE MATERIAL PROPERTIES
MP,NUXY,2,0.3                       *      (ISOTROPIC)
C***.....................................................................***
C*** GENERATE NODES AND ELEMENTS FOR BEAM MODEL                         ***
N,1    $N,13,12  $FILL              *NODES INCLUDE MIDSIDE NODES
NGEN,3,20,1,13,1,,1.5               *ONE ELEMENT ACROSS BEAM WIDTH
E,1,3,43,41,2,23,42,21              *SIX ELEMENTS ALONG BEAM LENGTH
EGEN,6,2,-1
ESEL,,6                             *SELECT END ELEMENT AND CONVERT
TYPE,2                              *TO TYPE 2 FOR PRINTOUT SELECTION
EMOD,ALL
EALL
C***.....................................................................***
C*** APPLY BOUNDARY CONDITIONS                                          ***
NSEL,X                              *FIX ONE END (ALL DOFS)
D,ALL,ALL
NSEL,X,12                           *IMPOSED DISPLACEMENT OF UZ=0.5
D,ALL,UZ,0.5                        *AT FREE END (ALL NODES)
NALL
ITER,1,1,1
PRDI,-1                             *NO DISPLACEMENT PRINTOUT
KRF,-1                              *NO REACTION FORCE PRINTOUT
C***.....................................................................***
C*** PLOT MODEL GEOMETRY AND BOUNDARY CONDITIONS                        ***
/SHOW,FILE33,,1
/VIEW,1,1,1,1
/VUP,1,Z
/PBC,ALL,1                          *PLOT ALL BOUNDARY CONDITIONS
/TITLE, MODEL 1: SANDWICH BEAM USING STIF99 ELEMENTS
EPLOT                               *PLOT ELEMENTS
AFWRITE
FINI
C***.....................................................................***
C*** SOLUTION PHASE                                                     ***
```

```
/INP,27
FINI
C***.....................................................***
C*** POSTPROCESS RESULTS                                  ***
/POST1
SET,1,1
/VIEW,1,1,1,1                    *SETUP VIEW DIRECTION AND ANGLE
/VUP,1,Z
/DSC,1,2                         *SCALE DEFORMATIONS X 2
/TITLE, MODEL 1 (STIF99) DEFORMATIONS
PLDISP,2                         *PLOT BEAM DEFORMATIONS
FINI
C*************************************************************************
C***                                                       ***
/PREP7
C***=====================================================***
C*** M O D E L   2 : 6 STIF46 ELEMENTS ALONG BEAM LENGTH   ***
C***                16 LAYERS PER ELEMENT                 ***
C***=====================================================***
/NOPR
ET,1,46,,,,,1,,1                 *TYPE1 SUPPRESS PRINTOUT
ET,2,46,,,,,1,4                  *TYPE2 FOR PRINTOUT; STORE STRESSES
C***.....................................................***
C*** REAL CONSTANTS - DEFINE SANDWICH STACKING SEQUENCE    ***
R,1,16,1                         *16 SYMMETRIC LAYERS
RMORE                            *(45/45/-45/-45/0/0/0/0)SYMM
RMORE,1, 45,.125, 1, 45,.125     *EACH OF TK=0.125
RMORE,1,-45,.125, 1,-45,.125     *(ONLY 8 ARE SPECIFIED - LSYM=1)
RMORE,2,,.125,2,,.125
RP2
C***.....................................................***
C*** DEFINE MATERIAL PROPERTIES FOR PLY AND CORE           ***
MP,EX,1,30E6                     *MAT1: PLY MATERIAL PROPERTIES
MP,EY,1,12E6                     *      (ORTHOTROPIC)
MP,NUXY,1,0.12
MP,GXY,1,7E6
MP,EX,2,3E6                      *MAT2: CORE MATERIAL PROPERTIES
MP,NUXY,2,0.3                    *      (ISOTROPIC)
C***.....................................................***
C*** GENERATE NODES AND ELEMENTS FOR BEAM MODEL            ***
N,1   $N,7,12  $FILL
NGEN,2,10,1,7,1,,3
NGEN,2,20,1,17,1,,,2
E,1,2,12,11,21,22,32,31          *ONE ELEMENT ACROSS BEAM WIDTH
EGEN,6,1,-1                      *SIX ELEMENTS ALONG BEAM LENGTH
ESEL,,6                          *CONVERT END ELEMENT TO TYPE2
TYPE,2
EMOD,ALL
EALL
C***.....................................................***
C*** APPLY BOUNDARY CONDITIONS                             ***
NSEL,X                           *FIX ALL DOF AT INE END
D,ALL,ALL
```

```
NSEL,X,12
D,ALL,UZ,0.5                        *IMPOSED DISPLACEMENT UZ=0.5 AT
NALL                                *FREE END
ITER,1,1
PRDI,-1
KRF,-1
C***.....................................................................***
C*** PLOT MODEL GEOMETRY AND BOUNDARY CONDITIONS                         ***
/VIEW,1,1,1,1
/VUP,1,Z
/TYPE,1,2
/TITLE, MODEL 2 : SANDWICH BEAM USING STIF46 ELEMENTS (UNSTACKED)
EPLOT                               *PLOT ELEMENTS
AFWRITE
FINI
C***.....................................................................***
C*** SOLUTION PHASE                                                      ***
/INP,27
FINI
C***.....................................................................***
C*** POSTPROCESS RESULTS                                                 ***
/POST1
SET,1,1
/VIEW,1,1,1,1                       *SETUP VIEW DIRECTION AND ANGLE
/VUP,1,Z
/DSC,1,2                            *SCALE DEFORMATIONS X 2
/TYPE,1,2
/TITLE, MODEL 2 (STIF46) DEFORMATIONS
PLDISP,2                            *PLOT BEAM DEFORMATIONS
FINI
C*************************************************************************
C***                                                                      ***
/PREP7
C***=====================================================================***
C*** M O D E L   3 : 6 STIF46 ELEMENTS ALONG BEAM LENGTH                  ***
C***                16 ELEMENTS THROUGH BEAM THICKNESS                   ***
C***                1 LAYER PER ELEMENT                                  ***
C***=====================================================================***
/NOPR
ET,1,46,,,,,1,,1                    *TYPE1 SUPPRESS PRINTOUT
ET,2,46,,,,,1,4                     *TYPE2 FOR PRINTOUT; STORE STRESSES
C***.....................................................................***
C*** REAL CONSTANTS - DEFINE PLY ANGLES                                  ***
R,1,1                               *45 DEGREES LAYER
RMORE
RMORE,1, 45,1
R,2,1                               *-45 DEGREES LAYER
RMORE
RMORE,1,-45,1
R,3,1                               *CORE; THICKNESSES ARE RELATIVE
RMORE
RMORE,2,    ,1
C***.....................................................................***
```

```
C*** DEFINE MATERIAL PROPERTIES FOR PLY AND CORE                    ***
MP,EX,1,30E6                        *MAT1: PLY MATERIAL PROPERTIES
MP,EY,1,12E6                        *      (ORTHOTROPIC)
MP,NUXY,1,0.12
MP,GXY,1,7E6
MP,EX,2,3E6                         *MAT2: CORE MATERIAL PROPERTIES
MP,NUXY,2,0.3                       *      (ISOTROPIC)
C***.....................................................................***
C*** GENERATE NODES AND ELEMENTS FOR BEAM MODEL                     ***
N,1    $N,7,12  $FILL
NGEN,2,10,1,7,1,,3
NGEN,17,20,1,17,1,,,0.125
REAL,1
E,1,2,12,11,21,22,32,31             *GENERATE STACK OF 16 ELEMENTS
EGEN,2,20,-1                        *ELEM 1 & 2 : 45/45     (REAL 1)
EGEN,2,40,-2,,,,,1                  *ELEM 3 & 4 : -45/-45   (REAL 2)
EGEN,2,40,-2,,,,,1                  *ELEM 5 TO 12 : 0/0     (REAL 3)
EGEN,4,40,-2
EGEN,2,40,-2,,,,,-1                 *ELEM 13 & 14 : -45/-45 (REAL 2)
EGEN,2,40,-2,,,,,-1                 *ELEM 15 & 16 : 45/45   (REAL 1)
EGEN,5,1,-16                        *GENERATE ELEMENTS ALONG LENGTH
EGEN,2,1,-16,,,,1                   *END STACK OF ELEMENTS IS TYPE 2
C***.....................................................................***
C*** APPLY BOUNDARY CONDITIONS                                      ***
NSEL,X                              *FIX ALL DOF AT INE END
D,ALL,ALL
NSEL,X,12
D,ALL,UZ,0.5                        *IMPOSED DISPLACEMENT UZ=0.5 AT
NALL                                *FREE END
ITER,1,1
PRDI,-1
KRF,-1
C***.....................................................................***
C*** PLOT MODEL GEOMETRY AND BOUNDARY CONDITIONS                    ***
/VIEW,1,1,1,1
/VUP,1,Z
/TYPE,1,2
/TITLE, MODEL 3 : SANDWICH BEAM USING STIF46 STACKED ELEMENTS
EPLOT                               *PLOT ELEMENTS
AFWRITE
FINI
C***.....................................................................***
C*** SOLUTION PHASE                                                ***
/INP,27
FINI
C***.....................................................................***
C*** POSTPROCESS RESULTS                                           ***
/POST1
SET,1,1
/VIEW,1,1,1,1                       *SETUP VIEW DIRECTION AND ANGLE
/VUP,1,Z
/DSC,1,2                            *SCALE DEFORMATIONS X 2
/TYPE,1,2
```

```
/TITLE, MODEL 3 (STIF46) DEFORMATIONS
PLDISP,2                          *PLOT BEAM DEFORMATIONS
ESEL,,49,96
/FOCUS,1,10,1,1.3
/DIST,1,3
/TITLE, MODEL 3: SXZ STRESSES AT FREE END
PLNSTR,SXZ
/TITLE, MODEL 3: SYZ STRESSES AT FREE END
PLNSTR,SYZ
EALL
/VIEW
/VUP
/AUTO
/TITLE, MODEL 3: SXZ STRESSES THROUGH BEAM THICKNESS (Y=0)
PLPATH,7,327,SXZ
/TITLE, MODEL 3: SYZ STRESSES THROUGH BEAM THICKNESS (Y=0)
PLPATH,7,327,SYZ
/TITLE, MODEL 3: SXZ STRESSES THROUGH BEAM THICKNESS (Y=3)
PLPATH,17,337,SXZ
/TITLE, MODEL 3: SYZ STRESSES THROUGH BEAM THICKNESS (Y=3)
PLPATH,17,337,SYZ
FINI
```

7.3 Thick – Walled Pressure Vessel Failure Optim. under internal pressure

```
C*************************************************************************
C*** EXAMPLE 3 : OPTIMUM ANGLE-PLY LAYUP AND THROUGH THICKNESS      ***
C***             STRESSES IN A PRESSURE VESSEL USING STIF46         ***
C*************************************************************************
C***                                                                 ***
C***.................................................................***
C*** DEFINE CONSTANT PARAMETERS                                      ***
THK=2                             *CYLINDER WALL THICKNESS
R0=10                             *INNER RADIUS
LEN=2                             *AXIAL ELEMENT LENGTH (ARBITRARY)
PRES=0.5                          *INTERNAL PRESSURE
PI=3.14159                        *VALUE OF PI RADIANS
C***.................................................................***
C*** CALCULATE DEPENDENT GEOMETRIC PARAMETERS                        ***
R1=(R0+THK)                       *OUTER RADIUS
CAPS=(PI*(R0*R0))                 *INNER X-SECTN AREA
AREA=(PI*((R1*R1)-(R0*R0)))       *OUTER X-SECTN AREA
F=((CAPS*PRES)/AREA)              *AXIAL PRESSURE DUE TO END CAPS
C***.................................................................***
C*** INITIALIZE VARIABLE PARAMETER (DESIGN VARIABLE)                 ***
ANG=45                            *STARTING WINDING ANGLE
C***.................................................................***
C*** OPTIMIZATION LOOP BEGINS HERE                                   ***
C***.................................................................***
C*** P R E - P R O C E S S I N G                                     ***
```

```
/PREP7
/TITLE, PART 1: OPTIMUM WINDING ANGLE USING SINGLE ANGLE-PLY
/OUT,26                          *DIVERT UNWANTED OUTPUT TO FILE26
/NOPR
ET,1,46,,,,,1,4,1
C***.....................................................................***
C*** STACKING SEQUENCE (REAL CONSTANTS)                                  ***
R,1,2
RMORE                            *1 UNIT OF (+ANG/-ANG) PLIES
RMORE,1,ANG,1,1,-ANG,1
MP,EX,1,181E3                    *ORTHOTROPIC MATERIAL PROPERTIES
MP,EY,1,103E3
MP,PRXY,1,.28
MP,PRYZ,1,0
MP,PRXZ,1,0
MP,GXY,1,7.17E3
C***.....................................................................***
C*** NON-LINEAR MATERIAL TABLE FOR FAILURE CRITERIA                      ***
NL,1,85,,,1              *USE TSAI-WU FAILURE CRITERIA
NL,1,91,0,0
NL,1,97,9999             * NON-TEMPERATURE DEPENDENT FAILURE CRITERIA
NL,1,157,1500              *FAILURE STRESS, X  DIRECTION TENSION
NL,1,163,-1500             *FAILURE STRESS, X  DIRECTION COMPRESSION
NL,1,169,40                *FAILURE STRESS, Y  DIRECTION TENSION
NL,1,175,-246              *FAILURE STRESS, Y  DIRECTION COMPRESSION
NL,1,181,40                *FAILURE STRESS, Z  DIRECTION TENSION
NL,1,187,-246              *FAILURE STRESS, Z  DIRECTION COMPRESSION
NL,1,193,68                *FAILURE STRESS, XY  DIRECTION SHEAR
NL,1,199,1000              *LARGE NUMBER FOR YZ DIRECTION SHEAR
NL,1,205,1000              *LARGE NUMBER FOR XZ DIRECTION SHEAR
NL,1,211,-1                *XY TSAI-WU COUPLING COEFFICIENT
NL,1,217,-1                *YZ TSAI-WU COUPLING COEFFICIENT
NL,1,223,-1                *XZ TSAI-WU COUPLING COEFFICIENT
C***.....................................................................***
C*** GENERATE MODEL IN CYLINDRICAL COORDINATES                           ***
CSYS,1                     *CYLINDRICAL COORDINATES
N,1,R0                     *1 ELEMENT THROUGH WALL THICKNESS
N,2,(R0+THK)
N,3,(R0+THK),,LEN
N,4,R0,,LEN
NGEN,9,4,1,4,1,,11.25
NROTAT,ALL
E,5,8,4,1,6,7,3,2          *8 ELEMENTS ALONG QUADRANT
EGEN,8,4,-1
C***.....................................................................***
C*** APPLY BOUNDARY CONDITIONS                                           ***
D,ALL,UY                   *SYMMETRY BOUNDARY CONDITIONS
SYMBC,1,3
EP,ALL,1,PRES              *APPLY INTERNAL PRESSURE
EP,ALL,3,-F                *AXIAL FORCE DUE TO END CAPS
NSEL,Z,LEN                 *COUPLE NODE ON SECTION ALONG AXIS
CPSIZE,50
CP,1,UZ,ALL
```

```
NALL
C***.....***
C*** PLOT MODEL GEOMETRY
/SHOW,FILE33,,1
/VIEW,1,-1,.2,1.5
/TYPE,,2
/PBC,PRES,2
EPLOT
C***.....***
ITER,1,1,1
AFWR
FINI
C***.....***
C*** S O L U T I O N   P H A S E                                    ***
/INP,27
FINI
C***.....***
C*** P O S T - P R O C E S S I N G                                  ***
/POST1
STRESS,FC3,46,58
SET,1                                  *FVAL = MAXIMUM VALUE OF
*GET,FVAL,FC3,1                       *       TSAI-WU FAILURE CRITERIA
*GET,UOUT,UX,2                        *UOUT = OUTER RADIAL DISPLACEMENT
*GET,UIN,UX,1                         *UIN = INNER RADIAL DISPLACEMENT
FINI
C***.....***
C*** O P T I M I Z A T I O N   M O D U L E                          ***
/OPT
OPSEL,8                              *SELECT 8 BEST DESIGN SETS
OPVAR,ANG,DV,0,90,.01                *ANG IS THE DESIGN VARIABLE
OPVAR,UOUT,SV,-1,1,1E-5
OPVAR,UIN,SV,-1,1,1E-5
OPVAR,FVAL,OBJ,,,1E-6                *FVAL IS THE OBJECTIVE FUNCTION
OPCOPY
/OUT
OPRUN,30,.6,,7                       *SELECT 7 RANDOM SOLUTIONS
OPLIST,ALL
C***.....***
C*** DISPLAY RESULTS
OPSORT,ANG                           *SORT LOOPS BY INCREASING ANGLE
XVAR,ANG                             *X-AXIS=ANGLE
/TITLE, VARIATION OF TSAI-WU CRITERIA WITH WINDING ANGLE
PLVAR,FVAL                           *FAILURE CRITERIA VERSUS ANGLE
/TITLE, VARIATION OF OUTER RADIAL DISPLACEMENT WITH WINDING ANGLE
PLVAR,UOUT                           *OUTER RADIAL DISP VERSUS ANGLE
FINI
C*************************************************************************
```

```
C*************************************************************************
C***                                                                   ***
ANG=56.1                             *OPTIMUM ANGLE FROM PART 1
```

```
C***.....***
C*** P A R T  2  :  P R E - P R O C E S S I N G                    ***
/PREP7
/TITLE, PART 2: VARIATION OF DEFORMATIONS AND STRESSES
/NOPR
C***.....***
C*** CALCULATE DEPENDENT GEOMETRIC PARAMETERS                     ***
ET,1,46,,,,,1,,1
C***.....***
C*** STACKING SEQUENCE (REAL CONSTANTS)                           ***
R,1,1
RMORE                                *1 LAYER OF (+ANG) PLY
RMORE,1,ANG,1
R,2,1                                *1 LAYER OF (-ANG) PLY
RMORE
RMORE,1,-ANG,1
C***.....***
C*** MATERIAL PROPERTIES                                          ***
MP,EX,1,181E3                        *SAME AS IN PART 1
MP,EY,1,103E3
MP,PRXY,1,.28
MP,PRYZ,1,0
MP,PRXZ,1,0
MP,GXY,1,7.17E3
C***.....***
C*** GENERATE MODEL IN CYLINDRICAL COORDINATES                    ***
CSYS,1
N,1,R0                               *QUARTER SYMMETRY MODEL USING
N,9,R0,90                            *8 ELEMENTS ALONG CIRCUMFERENCE
FILL
NGEN,2,10,1,9,1,,,LEN                *6 ELEMENTS THROUGH THE THICKNESS
NGEN,7,20,1,19,1,(THK/6)             *INNER 24 ELEMENTS HAVE REAL=1
NROTAT,ALL                           *OUTER 24 ELEMENTS HAVE REAL=2
E,2,12,11,1,22,32,31,21              *CORRESPONDING TO OPTIMUM
EGEN,8,1,-1                          *WINDING ANGLE (+55.6/-55.6)
EGEN,3,20,-8
EGEN,2,60,-24,,,,,1
C***.....***
C*** APPLY SYMMETRY BOUNDARY CONDITIONS AND PRESSURES
SYMBC,1,3
D,ALL,UY
ESEL,,1,8,1
EP,ALL,1,PRES
EALL
EP,ALL,3,-F
NSEL,Z,LEN
CPSIZE,65                            *INCREASE SIZE TO INCLUDE
CP,1,UZ,ALL                          *ALL X-SECTION NODES
NALL
C***.....***
C*** PLOT MODEL                                                   ***
/VIEW,1,-1,.2,1.5
/TYPE,,2
```

```
/PBC,PRES,2
EPLOT
C***.....................................................***
ITER,1,1,1
PRDIS,-1
PRRF,-1
AFWR
FINI
C***.....................................................***
C***  S O L U T I O N   P H A S E                        ***
/INP,27
FINI
C***.....................................................***
C***  P O S T - P R O C E S S I N G                      ***
/POST1
SET,1,1
CSYS,1
/TITLE, VARIATION OF RADIAL DISPLACEMENT ACROSS WALL
PLPA,1,121,UX
/TITLE, VARIATION OF RADIAL STRESS ACROSS WALL
PLPA,1,121,SX
/TITLE, VARIATION OF HOOP STRESS ACROSS WALL
PLPA,1,121,SY
/TITLE, VARIATION OF AXIAL STRESS ACROSS WALL
PLPA,1,121,SZ
/FOCU,1,8,8
/DIS,1,3
/TITLE, SX DISTRIBUTION THROUGH WALL
PLNS,SX
/TITLE, SY DISTRIBUTION THROUGH WALL
PLNS,SY
/TITLE, SZ DISTRIBUTION THROUGH WALL
PLNS,SZ
FINI
C*************************************************************************
```

7.4 Buckling Analysis of Compos.Cylind. Panel with Circular Hole

```
C*************************************************************************
C*** EXAMPLE 4: COMPOSITE CYLINDRICAL PANEL WITH CIRCULAR HOLE     ***
C***            USING STIF91 - 8 NODE LAYERED SHELL ELEMENT       ***
C*************************************************************************
/SHOW,FILE33,,1
/PREP7
C***=====================================================================
/TITLE, CYLINDRICAL PANEL : PHASE 1 - PRE-BUCKLING STATIC ANALYSIS
C***=====================================================================
ET,1,91,,,16                          *16 LAYERS
C***.....................................................
C*** REAL CONSTANTS DEFINE LAYER MATERIAL, ANGLE, THICKNESS, STACKING
C*** (45/-45/90/0/0/90/-45/45) SYMM., PLY THK=0.00567
```

```
R,1  ,1, 45, 0.0056     *LAYER 1      (BOTTOM TO TOP)
RMORE,1,-45, 0.0056     *LAYER 2
RMORE,1, 90, 0.0056     *LAYER 3
RMORE,1,  0, 0.0056     *LAYER 4
RMORE,1,  0, 0.0056     *LAYER 5
RMORE,1, 90, 0.0056     *LAYER 6
RMORE,1,-45, 0.0056     *LAYER 7
RMORE,1, 45, 0.0056     *LAYER 8
RMORE,1, 45, 0.0056     *LAYER 9
RMORE,1,-45, 0.0056     *LAYER 10
RMORE,1, 90, 0.0056     *LAYER 11
RMORE,1,  0, 0.0056     *LAYER 12
RMORE,1,  0, 0.0056     *LAYER 13
RMORE,1, 90, 0.0056     *LAYER 14
RMORE,1,-45, 0.0056     *LAYER 15
RMORE,1, 45, 0.0056     *LAYER 16
C***.....................................................................
C*** DEFINE ORTHOTROPIC MATERIAL PROPERTIES - MATERIAL #1
MP,EX,  1,19.6E6                *ORTHOTROPIC MATERIAL PROPERTIES
MP,EY,  1,1.89E6                *EACH PLY USES SAME MATERIAL
MP,NUXY,1,0.0376                *HOWEVER FIBER DIRECTIONS DIFFER
MP,GXY, 1,0.93E6
MP,GXZ, 1,0.93E6
MP,GYZ, 1,0.63E6
C***.....................................................................
C*** MESH MODULE : GENERATE KEYPOINTS AND AREAS IN CYLINDRICAL SYSTEM
CSYS,1
K,1,15                          *DEFINE KEYPOINTS AT CORNERS OF
K,2,15,27.818                   *A QUADRANT OF THE PANEL OF
K,3,15,27.818,7                 *RADIUS=15.0; ARC ANGLE=55.6
K,4,15,,7
A,1,2,3,4                       *AREA #1 QUARTER PANEL WITHOUT HOLE
LOCAL,11,1,10,,,90,,90          *DEFINE A LOCAL CS TO GENERATE
CSYS,11                         *AN INTERSECTING CYLINDER
K,101,1                         *HOLE RADIUS=1.0
K,102,1,90
K,103,1,90,10
K,104,1,,10
A,101,102,103,104               *AREA #2 IS A SCRATCH CYLINDER
LAINT,1,2                       *USED TO GENERATE THE HOLE BOUNDARY
LDIV,9                          *DIVIDE HOLE BOUNDARY INTO 2 LINES
ADEL,1,2                        *DELETE SCRATCH AREAS
CSYS,1                          *DEFINE 2 AREAS IN QUARTER PANEL
L,6,4,6,2.5                     *DEFINE LINE DIVISIONS WITH SPACING
L,7,3,6,2.5                     *RATIO TO BIAS MESH TOWARDS HOLE
L,5,2,6,2.5
A,7,5,2,3                       *IN CSYS 1 AND SEQUENCE NUMBERING
A,6,7,3,4
NUMCMP,AREA
ELSIZE,,2                       *2 ELEMENT DIVISIONS FOR MESHING
LOCAL,12,1                      *DEFINE CYL. CS AS REF FOR ESYS
ESYS,12                         *DEFINE ESYS TO ORIENT ALL ELEMENTS
AMESH,ALL                       *MESH QUARTER PANEL AREAS (STIF91)
```

```
ARSYM,2,1,2                          *REFLECT PANEL - GET TOP HALF
CSYS,11                              *CHANGE CSYS FOR 2ND REFLECTION
ARSYM,2,1,4                          *REFLECT AGAIN; GET FULL MESHED PANEL
NUMMRG,ALL                           *MERGE AND RENUMBER NODES
NUMCMP,ALL
CSYS,1
NROT,ALL                             *ROTATE NODAL C.S. TO CSYS 1
C***.....................................................................
C*** BOUNDARY CONDITIONS SPECIFICATION
NSEL,Z,-7                            *SELECT BOTTOM CURVED EDGE
D,ALL,ALL                            *CLAMP ALL DISPLACEMENTS
NSEL,Z,7                             *SELECT TOP CURVED EDGE
D,ALL,UX,,,,,UY,ROTX,ROTY,ROTZ       *CLAMP ALL BUT AXIAL DISPLACEMENTS
D,ALL,UZ,-0.0316
NSEL,Y,-27.818                       *SIMPLY SUPPORTED STRAIGHT EDGES
D,ALL,UX,,,,,ROTY
NSEL,Y,27.818
D,ALL,UX,,,,,ROTY
NALL
C***.....................................................................
C*** PLOT MODEL GEOMETRY AND BOUNDARY CONDITIONS APPLIED
/VIEW,1,1,-0.5,0.7                   *SET UP VIEW PARAMETERS
/VUP,1,Z
/PSYM,ESYS,1                         *SHOW ESYS ORIENTATION
EPLOT                                *PLOT ELEMENTS
/PBC,ALL,1                           *PLOT ALL BOUNDARY CONDITIONS
NPLOT
ITER,1,,1                            *SUPPRESS SOLUTION PRINTOUT
AFWRIT
FINI
/INP,27
FINI
C***.....................................................................
C*** POST-PROCESS RESULTS: DEFORMATIONS AND EDGE DISPLACEMENTS
/POST1
SET,1,1
CSYS,1
/VIEW,1,1,-0.5,0.7                   *SET UP VIEW PARAMETERS
/VUP,1,Z
/DSC,1,35
PLDI,2                               *PLOT DEFORMATIONS
TOP
/VIEW,1,1
/VUP,1,Z
/DSC
/TITLE, SZ STRESS DISTRIBUTION (TOP)
PLNSTR,SZ                            *NODAL STRESS SZ (TOP) IN CSYS 1
PRNSTR,COMP
FINI
C***=====================================================================
/TITLE, CYLINDRICAL PANEL : PHASE 2 - BUCKLING ANALYSIS
C***=====================================================================
/BUCKLE,35,5,5                       *SPECIFY TOTAL MASTER DOFS
```

```
ITER,1,1,1                          *SUB SPACE ITERATION
END
FINI
C***.....................................................................
C*** POST PROCESS BUCKLING MODE SHAPES FOR FIRST THREE MODES
/POST1
SET,1,1                             *FIRST BUCKLING MODE
/VIEW,1,0.5,-0.5,1                  *SET UP VIEW PARAMETERS
/VUP,1,X
/TYPE,1,2
/VCONE,1,20
/DIST,1,30
/DSC,1,1
/TITLE, PHASE 2 - BUCKLING ANALYSIS, FIRST MODE
PLDISP
C***.....................................................................
SET,1,2                             *SECOND BUCKLED MODE
/TITLE, PHASE 2 - BUCKLING ANALYSIS, SECOND MODE
PLDISP
C***.....................................................................
SET,1,3                             *THIRD BUCKLED MODE
/TITLE, PHASE 2 - BUCKLING ANALYSIS, THIRD MODE
PLDISP
FINI
C*************************************************************************
```

7.5 Documentation for Macro File LAYUP

```
c*************************************************************************
c***
c***        This layer stacking user file can be used to generate
c***        real constants required for composite elements STIF46 and STIF99.
c***
c***        usage:  *use,layup,ARG1,ARG2,ARG3,ARG4,ARG5,ARG6,ARG7,ARG8,ARG9
c***
c***              where: ARG1 - real constant number
c***                     ARG2 - material number
c***                     ARG3 - layer thickness
c***                     ARG4  - orientation angle for layer 1
c***                     ARG5  -      "         "    "    "   2
c***                     ARG6  -      "         "    "    "   3
c***                     ARG7  -      "         "    "    "   4
c***                     ARG8 - symmetry field
c***                     ARG9 - repeat pattern field
c***
c***
c***              notes:
c***
c***                1. basic layup sequence limited to 4 layers.
c***                   at least 1 layer is assumed.
c***
c***                2. layers not used should have argx = 999
```

```
c***
c***                  first occurrence of argx = 999 will ignore
c***                  remaining argx values for layer angles
c***
c***                     example  ARG4 = 20, ARG5 = 999
c***                              1 layer assumed - angle = 20 degrees
c***
c***                     example  ARG4 = 20, ARG5 = 10, ARG6 = 999
c***                              ARG7 = 999
c***                              2 layers - angles 20 and 10 degrees
c***
c***                     example  ARG4 = 999, ARG5 = 10, ARG6 = 0 = ARG7
c***                              4 layers - angles 999,10,0,0
c***                              note - ARG4 is always used
c***
c***               3. ARG8 = 1 - symmetric layup
c***
c***                     example  ARG4 = -45, ARG5 = 60
c***                              layup will be -45/60/60/-45
c***
c***                  ARG8 = -1 - antisymmetric layup
c***
c***                     example  ARG4 = -45, ARG5 = 60
c***                              layup will be -45/60/-60/45
c***
c***                     example  ARG4 =   0, ARG5 = 60
c***                              layup will be 0/60/-60/90
c***                              note - if ARG4,ARG5,ARG6,ARG7 = 0/90
c***                              antisym layer will be 90/0
c***
c***               4. ARG9 will generate "n" repetitions of the
c***                  basic layup.
c***
c***               5. ARG8 and ARG9 are exclusive, i.e. if ARG9 is
c***                  positive ARG8 is ignorred.
c***
c*******************************************************************************
```

7.6 Macro File LAYUP

```
*STAT
*IF,ARG5,EQ,999,:LE1
*IF,ARG6,EQ,999,:LE2
*IF,ARG7,EQ,999,:LE3
NLAY=4
*IF,ARG9,GT,0,:R4
*IF,ARG8,LT,0,:S41
*IF,ARG8,GT,0,:S42
R,ARG1,NLAY
*GO,:J4
:S42
R,ARG1,NLAY*2,1
```

```
:J4
RMORE
RMORE,ARG2,ARG4,ARG3,ARG2,ARG5,ARG3
RMORE,ARG2,ARG6,ARG3,ARG2,ARG7,ARG3
*GO,:END
:S41
R,ARG1,NLAY*2
RMORE
RMORE,ARG2,ARG4,ARG3,ARG2,ARG5,ARG3
RMORE,ARG2,ARG6,ARG3,ARG2,ARG7,ARG3
RMORE,ARG2,-ARG7,ARG3,ARG2,-ARG6,ARG3
RMORE,ARG2,-ARG5,ARG3,ARG2,-ARG4,ARG3
*IF,ARG4,EQ,0,:4S1
*IF,ARG4,EQ,90,:4S2
*GO,:4J1
:4S1
RMOD,ARG1,35,90
*GO,:4J1
:4S2
RMOD,ARG1,35,0
:4J1
*IF,ARG5,EQ,0,:4S3
*IF,ARG5,EQ,90,:4S4
*GO,:4J2
:4S3
RMOD,ARG1,32,90
*GO,:4J2
:4S4
RMOD,ARG1,32,0
:4J2
*IF,ARG6,EQ,0,:4S5
*IF,ARG6,EQ,90,:4S6
*GO,:4J3
:4S5
RMOD,ARG1,29,90
*GO,:4J3
:4S6
RMOD,ARG1,29,0
:4J3
*IF,ARG7,EQ,0,:4S7
*IF,ARG7,EQ,90,:4S8
*GO,:4J4
:4S7
RMOD,ARG1,26,90
*GO,:4J4
:4S8
RMOD,ARG1,26,0
:4J4
*GO,:END
:R4
R,ARG1,NLAY*ARG9
RMORE
N=0
```

```
:RP4
N=N+1
RMOD,ARG1,(1+(12*N)),ARG2,ARG4,ARG3
RMOD,ARG1,(4+(12*N)),ARG2,ARG5,ARG3
RMOD,ARG1,(7+(12*N)),ARG2,ARG6,ARG3
RMOD,ARG1,(10+(12*N)),ARG2,ARG7,ARG3
*IF,N,LT,ARG9,:RP4
*GO,:END
:LE1
NLAY=1
*IF,ARG9,GT,0,:R1
*IF,ARG8,LT,0,:S11
*IF,ARG8,GT,0,:S12
R,ARG1,NLAY
*GO,:J1
:S12
R,ARG1,NLAY*2,1
:J1
RMORE
RMORE,ARG2,ARG4,ARG3
*GO,:END
:S11
R,ARG1,NLAY*2
RMORE
RMORE,ARG2,ARG4,ARG3,ARG2,-ARG4,ARG3
*IF,ARG4,EQ,0,:1S1
*IF,ARG4,EQ,90,:1S2
*GO,:1J1
:1S1
RMOD,ARG1,17,90
*GO,:1J1
:1S2
RMOD,ARG1,17,0
:1J1
*GO,:END
:R1
R,ARG1,NLAY*ARG9
RMORE
N=0
:RP1
N=N+1
RMOD,ARG1,(10+(3*N)),ARG2,ARG4,ARG3
*IF,N,LT,ARG9,:RP1
*GO,:END
:LE2
NLAY=2
*IF,ARG9,GT,0,:R2
*IF,ARG8,LT,0,:S21
*IF,ARG8,GT,0,:S22
R,ARG1,NLAY
*GO,:J2
:S22
R,ARG1,NLAY*2,1
```

```
:J2
RMORE
RMORE,ARG2,ARG4,ARG3,ARG2,ARG5,ARG3
*GO,:END
:S21
R,ARG1,NLAY*2
RMORE
RMORE,ARG2,ARG4,ARG3,ARG2,ARG5,ARG3
RMORE,ARG2,-ARG5,ARG3,ARG2,-ARG4,ARG3
*IF,ARG4,EQ,0,:2S1
*IF,ARG4,EQ,90,:2S2
*GO,:2J1
:2S1
RMOD,ARG1,23,90
*GO,:2J1
:2S2
RMOD,ARG1,23,0
:2J1
*IF,ARG5,EQ,0,:2S3
*IF,ARG5,EQ,90,:2S4
*GO,:2J2
:2S3
RMOD,ARG1,20,90
*GO,:2J2
:2S4
RMOD,ARG1,20,0
:2J2
*GO,:END
:R2
R,ARG1,NLAY*ARG9
RMORE
N=0
:RP2
N=N+1
RMOD,ARG1,(7+(6*N)),ARG2,ARG4,ARG3
RMOD,ARG1,(10+(6*N)),ARG2,ARG5,ARG3
*IF,N,LT,ARG9,:RP2
*GO,:END
:LE3
NLAY=3
*IF,ARG9,GT,0,:R3
*IF,ARG8,LT,0,:S31
*IF,ARG8,GT,0,:S32
R,ARG1,NLAY
*GO,:J3
:S32
R,ARG1,NLAY*2,1
:J3
RMORE
RMORE,ARG2,ARG4,ARG3,ARG2,ARG5,ARG3
RMORE,ARG2,ARG6,ARG3
*GO,:END
:S31
```

```
R,ARG1,NLAY*2
RMORE
RMORE,ARG2,ARG4,ARG3,ARG2,ARG5,ARG3
RMORE,ARG2,ARG6,ARG3,ARG2,-ARG6,ARG3
RMORE,ARG2,-ARG5,ARG3,ARG2,-ARG4,ARG3
*IF,ARG4,EQ,0,:3S1
*IF,ARG4,EQ,90,:3S2
*GO,:3J1
:3S1
RMOD,ARG1,29,90
*GO,:3J1
:3S2
RMOD,ARG1,29,0
:3J1
*IF,ARG5,EQ,0,:3S3
*IF,ARG5,EQ,90,:3S4
*GO,:3J2
:3S3
RMOD,ARG1,26,90
*GO,:3J2
:3S4
RMOD,ARG1,26,0
:3J2
*IF,ARG6,EQ,0,:3S5
*IF,ARG6,EQ,90,:3S6
*GO,:3J3
:3S5
RMOD,ARG1,23,90
*GO,:3J3
:3S6
RMOD,ARG1,23,0
:3J3
*GO,:END
:R3
R,ARG1,NLAY*ARG9
RMORE
N=0
:RP3
N=N+1
RMOD,ARG1,(4+(9*N)),ARG2,ARG4,ARG3
RMOD,ARG1,(7+(9*N)),ARG2,ARG5,ARG3
RMOD,ARG1,(10+(9*N)),ARG2,ARG6,ARG3
*IF,N,LT,ARG9,:RP3
*GO,:END
:END
/GOPR
```

7.7 Documentation for Macro File COLIST

```
C*************************************************************************
C***    COLIST WILL GENERATE A COMPLETE LISTING
C***    OF ALL PERTINENT DATA FOR EVERY COMPOSITE
C***
C***        FOR EACH ELEMENT THE LISTING CONTAINS
C***             1.  ELIST - DISPLAYS ATTRIBUTES AND NODE NUMBERS
C***
C***             2.  RLIS  - LISTS THE LAYERS (MAT,ORIENTATION,THICKNESS)
C***
C***             3.  CSLIST - LIST OF THE BASE ELEMENT COORDINATE SYSTEM
C***                          IF IT IS SYSTEM 11 THRU 40
C***
C***    PRIOR TO USING COLIST, SELECT A SUBSET OF
C***    ELEMENTS FOR WHICH THE EXPANDED LISTING IS DESIRED
C***
C***    THIS SUBSET SHOULD BE COMPRISED OF ALL STIF91 ELEMENTS,
C***    ALL STIF46 ELEMENTS, OR ALL STIF99 ELEMENTS
C***
C***       ENTER *USE,COLIST,ARG1,ARG2,ARG3
C***
C***           WHERE:  ARG1 - MAXIMUM NUMBER OF LAYERS FOR
C***                          ALL ELEMENTS IN THE CURRENT SUBSET
C***
C***                   ARG2 - ENTER 0 IF CONSTANT THICKNESS FOR ALL
C***                          ELEMENTS IN THE CURRENT SUBSET
C***
C***                        - ENTER 1 IF TAPERED THICKNESS FOR ALL
C***                          ELEMENTS IN THE CURRENT SUBSET
C***
C***                   ARG3 - ENTER 91 IF THE ELEMENTS IN THE CURRENT
C***                          SUBSET CONSIST OF STIF91 COMPOSITES
C***
C***       NOTES:
C***              1. THE DATABASE WILL BE "SAVED" TO PREVENT LOSS
C***                 OF REAL CONSTANT DATA
C***
C***              2. THE REAL CONSTANT TABLE SIZE WILL BE RESET TO
C***                 MINIMIZE THE LISTING PER ELEMENT.
C***
C***              3. UPON COMPLETION OF THE LISTING THE DATABASE WILL
C***                 BE RESTORED USING "RESUME".
C***
C*************************************************************************
```

7.8 Macro File COLIST

```
SAVE
/NOPR
*IF,ARG3,EQ,91,:J91
*IF,ARG2,EQ,1:T99
RSIZE,(ARG1*3)+12
*GO,:CON
:T99
RSIZE,(ARG1*6)+12
*GO,:CON
:J91
RSIZE,(ARG1*6)
:CON
*GET,ELM,ELMN
*IF,ELM,LE,0,:JCP
*GET,RL,REAL,ELM
*GET,MT,MAT,ELM
*GET,ECS,ESYS,ELM
*IF,ECS,LT,11,:JDP
/GOPR
ELIS,ELM
RLIS,RL
CSLIS,ECS
/NOPR
EUSEL,,ELM
*GO,:CON
:JDP
/GOPR
ELIS,ELM
RLIS,RL
/NOPR
EUSEL,,ELM
*GO,:CON
:JCP
/GOPR
RESUME
```

8. Fracture Mechanics with ANSYS

8.1 Single – Edge Crack : Elastic Analysis

```
/PREP7
/TITLE, EX. 1-1: SINGLE-EDGE CRACK
C*************************** DEFINE PARAMETERS ************************************
A=1.0          * DEFINE CRACK LENGTH (IN)
B=1.0          * DEFINE HALF-WIDTH (IN)
B2=(2*B)       * FULL WIDTH (IN)
C=(6*B)        * DEFINE LENGTH (IN)
E=10E6         * DEFINE YOUNG'S MODULUS (PSI)
NU=0.333       * DEFINE POISSON'S RATIO
SIG=100        * DEFINE APPLIED LOAD (PSI)
C*************************** BUILD THE SOLID MODEL ********************************
K,1,-A,0   $K,2,0,0   $K,3,(B2-A),0      * KEYPOINTS
K,4,(B2-A),B   $K,5,(B2-A),C
K,6,-A,C   $K,7,-A,B
L,1,2    $RP6,1,1   $L,7,1   $L,4,7      * LINE SEGMENTS
AL,1,2,3,8,7                             * NEAR-FIELD AREA 1
AL,4,5,6,8                               * FAR-FIELD AREA 2
C*********************** BUILD THE FINITE ELEMENT MODEL ****************************
ET,1,2,,,2                * PLANE STRAIN 6-NODE TRIANGLE
MP,EX,1,E                 * MATERIAL PROPERTIES
MP,NUXY,1,NU
ELSIZE,B/4                * MAX ELEMENT SIZE IN THE NEAR FIELD
KSCON,2,B/18,1,8          * CRACK-TIP ELEMENTS
AMESH,1
ELSIZE,B/2                * MAX ELEMENT SIZE IN THE FAR FIELD
AMESH,2
WSORT,Y                   * REORDER MODEL
C************************ BOUNDARY CONDITIONS AND LOAD ****************************
ITER,1,0,1
KSE,1                                 * SAVE STRAIN ENERGY
NSEL,Y,0,0   $NRSEL,X,0,(10*B)        * SYMMETRY BOUNDARY CONDITION (Y=0,X>0)
D,ALL,UY,0
NRSEL,X,0,0                           * FIX ONE NODE IN X-DIRECTION (X=Y=0)
D,ALL,UX,0
NALL
PSF,0,2,C,-SIG                        * APPLY UNIFORM STRESS (Y=C)
AFWRITE
FINISH
/INPUT,27
FINISH
/POST1
C***************** IN POST1 USE KCALC COMMAND TO DETERMINE KI **********************
SET,1,1
NSEL,Y,0,0                        * SELECT NODES AT Y=0
NSORT,X
PRNODE                            * PRINT SELECTED RESULTS
PRDISP
PRNSTR,COMP
```

```
NALL
LPATH,2,19,18                    * SELECT NODES FOR KCALC COMMAND
KCALC,,,1                        * COMPUTE KI FOR A HALF-MODEL WITH SYM. B.C.
*GET,KI,KCAL,KI                  * GET KI AS A PARAMETER
CON1=((3.141593*A)**.5)          * CONSTANT = SQRT(PI*A)
KI=KI/(SIG*CON1)                 * MAKE KI DIMENSIONLESS
*STAT                            * VIEW RESULTS
C***************** IN POST1 DETERMINE KI FROM J-INTEGRAL **************************
RESET                            * START POSTPROCESSING AGAIN
SET,1,1
CON1=((3.141593*A)**.5)      * CONSTANT = SQRT(PI*A)
CON2=(E/(1-(NU*NU)))         * J-TO-K CONVERSION FACTOR (PLANE STRAIN)
LPATH,24,91,88,16            * DEFINE FIRST PATH
*USE,JIN2                    * USE THE JIN2 MACRO TO COMPUTE J
KI1=((CON2*JINT)**.5)        * CALCULATE K FROM J
KI1=KI1/(SIG*CON1)           * MAKE KI DIMENSIONLESS
LPATH,29,342,339,11          * SECOND PATH
*USE,JIN2
KI2=((CON2*JINT)**.5)
KI2=KI2/(SIG*CON1)
LPATH,32,345,125,8           * THIRD PATH
*USE,JIN2
KI3=((CON2*JINT)**.5)
KI3=KI3/(SIG*CON1)
LPATH,36,105,106,4           * FOURTH PATH
*USE,JIN2
KI4=((CON2*JINT)**.5)
KI4=KI4/(SIG*CON1)
*STAT                        * VIEW RESULTS
FINISH
```

8.2 Single – Edge Crack : Nodal Forces and Displacements

```
/PREP7
/TITLE, EX. 1-2: SINGLE-EDGE CRACK
C***************************** DEFINE PARAMETERS ************************************
A=1.0          * DEFINE CRACK LENGTH (IN)
B=1.0          * DEFINE HALF-WIDTH (IN)
B2=(2*B)       * FULL WIDTH (IN)
C=(6*B)        * DEFINE LENGTH (IN)
E=10E6         * DEFINE YOUNG'S MODULUS (PSI)
NU=0.333       * DEFINE POISSON'S RATIO
SIG=100        * DEFINE APPLIED LOAD (PSI)
C***************************** BUILD THE SOLID MODEL ********************************
K,1,-A,0   $K,2,0,0   $K,3,(B2-A),0      * KEYPOINTS
K,4,(B2-A),B   $K,5,(B2-A),C
K,6,-A,C   $K,7,-A,B
L,1,2    $RP6,1,1   $L,7,1   $L,4,7      * LINE SEGMENTS
AL,1,2,3,8,7                             * NEAR-FIELD AREA 1
AL,4,5,6,8                               * FAR-FIELD AREA 2
C*********************** BUILD THE FINITE ELEMENT MODEL ****************************
ET,1,2,,,2                * PLANE STRAIN 6-NODE TRIANGLE
```

```
MP,EX,1,E                  * MATERIAL PROPERTIES
MP,NUXY,1,NU
ELSIZE,B/4                 * MAX ELEMENT SIZE IN THE NEAR FIELD
KSCON,2,B/18,0,8           * CRACK-TIP ELEMENTS (NON-SINGULAR)
AMESH,1
ELSIZE,B/2                 * MAX ELEMENT SIZE IN THE FAR FIELD
AMESH,2
WSORT,Y                    * REORDER MODEL
C************************ BOUNDARY CONDITIONS AND LOAD ****************************
ITER,1,0,1
KSE,1                                  * SAVE STRAIN ENERGY
NSEL,Y,0,0   $NRSEL,X,0,(10*B)         * SYMMETRY BOUNDARY CONDITION (Y=0,X>0)
D,ALL,UY,0
NRSEL,X,0,0                            * FIX ONE NODE IN X-DIRECTION (X=Y=0)
D,ALL,UX,0
NALL
PSF,0,2,C,-SIG                         * APPLY UNIFORM STRESS (Y=C)
AFWRITE
FINISH
/INPUT,27
FINISH
/POST1
C****** POST1 LISTING OF NODAL FORCES AND DISPLACEMENTS NEAR THE CRACK TIP *******
SET,1,1
NSEL,Y,0  $NRSEL,X,0,B/17      * SELECT NODES AHEAD OF THE CRACK TIP
PRRFOR    $PRNODE              * LIST REACTION FORCES  AND COORDINATES
NSEL,Y,0  $NRSEL,X,-B/17,0     * SELECT NODES BEHIND THE CRACK TIP
PRDISP    $PRNODE              * LIST DISPLACEMENTS AND COORDINATES
C*** HAND CALCULATION:   G=(FY2*UY16 + FY19*UY17)/(1*B/18)
C***                     G=(46.220*.16278E-4 + 42.107*.11085E-4)/(1*1/18)
C***                     G=.0219443 IN-LB/IN*IN
C***                     KI=SQRT[E*G/(1-NU*NU)]/[SIG*SQRT(PI*A)]
C***                     KI=SQRT[10E6*.0219443/(1-.333*.333)]/[100*SQRT(PI*1)]
C***                     KI=2.80
FINISH
```

8.3 Single – Edge Crack : Energy Difference Virt. Crack Ext. KI Comp

```
/PREP7
/TITLE, EX. 1-3: SINGLE-EDGE CRACK
C***************************** DEFINE PARAMETERS ************************************
A=0.995       * DEFINE INITIAL CRACK LENGTH (IN)
B=1.0         * DEFINE HALF-WIDTH (IN)
B2=(2*B)      * FULL WIDTH (IN)
C=(6*B)       * DEFINE LENGTH (IN)
E=10E6        * DEFINE YOUNG'S MODULUS (PSI)
NU=0.333      * DEFINE POISSON'S RATIO
SIG=100       * DEFINE APPLIED LOAD (PSI)
C***************************** BUILD THE SOLID MODEL *********************************
K,1,-A,0   $K,2,0,0   $K,3,(B2-A),0     * KEYPOINTS
K,4,(B2-A),B   $K,5,(B2-A),C
K,6,-A,C   $K,7,-A,B
```

```
L,1,2    $RP6,1,1   $L,7,1   $L,4,7     * LINE SEGMENTS
AL,1,2,3,8,7                           * NEAR-FIELD AREA 1
AL,4,5,6,8                             * FAR-FIELD AREA 2
C********************** BUILD THE FINITE ELEMENT MODEL ****************************
ET,1,2,,,2               * PLANE STRAIN 6-NODE TRIANGLE
MP,EX,1,E                * MATERIAL PROPERTIES
MP,NUXY,1,NU
ELSIZE,B/2               * MAX ELEMENT SIZE IN THE NEAR FIELD
KSCON,2,B/9,1,4          * CRACK-TIP ELEMENTS
AMESH,1
ELSIZE,B                 * MAX ELEMENT SIZE IN THE FAR FIELD
AMESH,2
WSORT,Y                  * REORDER MODEL
C********************** BOUNDARY CONDITIONS AND LOAD ****************************
ITER,1,0,1
KSE,1                              * SAVE STRAIN ENERGY
NSEL,Y,0,0   $NRSEL,X,0,(10*B)     * SYMMETRY BOUNDARY CONDITION (Y=0,X>0)
D,ALL,UY,0
NRSEL,X,0,0                        * FIX ONE NODE IN X-DIRECTION (X=Y=0)
D,ALL,UX,0
NALL
PSF,0,2,C,-SIG                     * APPLY UNIFORM STRESS (Y=C)
AFWRITE
FINISH
C********************** FIRST ANALYSIS  -  A = A1 = 0.995 ****************************
/INPUT,27
FINISH
/POST1
C********************** IN POST1 CALCULATE STRAIN ENERGY ***************************
STRESS,SENE                  * STORE STRAIN ENERGY PER ELEMENT
SET,1,1
SSUM                         * SUM FOR TOTAL STRAIN ENERGY
*GET,SE1,SSUM,SENE           * GET TOTAL STRAIN ENERGY (SE1) FOR FIRST MODEL
SE1=(2*SE1)                  * DOUBLE FOR SYM. MODEL
FINISH
/PREP7
C********************** PERTURB MODEL FOR CRACK EXTENSION ****************************
RESUME,,,,1
A1=A                           * INITIAL CRACK LENGTH (IN)
A2=1.005                       * FINAL CRACK LENGTH (IN)
A=(0.5*(A1+A2))                * AVERAGE CRACK LENGTH
DA=(A2-A1)                     * CRACK EXTENSION
MODMESH,DETACH                 * DETACH SOLID AND FINITE ELEMENT MODELS
CSYS,1   $NSEL,X,0,A/2         * SELECT NODES SURROUNDING CRACK TIP (WITHIN A/2)
CSYS,0   $NGEN,2,0,ALL,,,DA    * EXTEND MODEL TO REPRESENT 1% CRACK EXTENSION
NALL
C********************** BOUNDARY CONDITIONS AND LOAD ****************************
ITER,1,0,1
KSE,1                              * SAVE STRAIN ENERGY
NSEL,Y,0,0   $NRSEL,X,DA,(10*B)    * SYMMETRY BOUNDARY CONDITION (Y=0,X>DA)
D,ALL,UY,0
NRSEL,X,DA,DA                      * FIX ONE NODE IN X-DIRECTION (X=Y=0)
D,ALL,UX,0
```

```
NALL
PSF,0,2,C,-SIG                    * APPLY UNIFORM STRESS (Y=C)
AFWRITE
FINISH
C******************** SECOND ANALYSIS  - A = A2 = 1.005 ***************************
/INPUT,27
FINISH
/POST1
C********************** IN POST1 CALCULATE STRAIN ENERGY ************************
STRESS,SENE                 * STORE STRAIN ENERGY PER ELEMENT
SET,1,1
SSUM                        * SUM FOR TOTAL STRAIN ENERGY
*GET,SE2,SSUM,SENE          * GET TOTAL STRAIN ENERGY (SE2) FOR SECOND MODEL
SE2=(2*SE2)                 * DOUBLE FOR SYM. MODEL
G=((SE2-SE1)/DA)            * CALCULATE ENERGY RELEASE RATE (AVG. A=1.00)
CON1=(E/(1-(NU*NU)))        * G-TO-KI CONVERSION FACTOR (PLANE STRAIN)
KI=(G*CON1)**.5             * CALCULATE KI FROM G (PLANE STRAIN CONDITIONS)
CON2=((3.141593*A)**.5)     * CONSTANT = SQRT(PI*A)
KI=KI/(SIG*CON2)            * DIMENSIONLESS KI
*STAT                       * VIEW RESULTS
FINISH
```

8.4 Inclined Edge Crack : Elastic Analysis

```
/PREP7
/TITLE, EX. 2: INCLINED EDGE CRACK
C******************************* DEFINE PARAMETERS ***********************************
A=1.0               * DEFINE CRACK LENGTH (IN)
W=(3.33333*A)       * DEFINE WIDTH (IN)
L=(16.66667*A)      * DEFINE LENGTH (IN)
L2=L/2              * DEFINE HALF LENGTH (IN)
SIG=100             * DEFINE TENSILE LOAD (PSI)
E=30E6              * DEFINE YOUNG'S MODULUS (PSI)
NU=0.3              * DEFINE POISSON'S RATIO
C************************** DEFINE MATERIAL PROPERTIES ******************************
MP,EX,1,E                       * YOUNGS'S MODULUS = 30E6 PSI
MP,NUXY,1,NU                    * POISSON'S RATIO = 0.3
C******************************** DEFINE SOLID MODEL ********************************
K,1,,(L2+(0.353553*A))                  * KEYPOINTS
K,3,,(L2+(0.353553*A))
K,2,(0.707107*A),(L2-(0.353553*A))
K,4  $K,5,W
K,6,W,(L2-(0.893934*W))
K,7,W,L  $K,8,,L
L,4,5  $L,5,6  $L,6,2  $L,2,1  $L,1,4   * LINES
L,6,7  $L,7,8  $L,8,3  $L,3,2
AL,1,2,3,4,5                            * AREAS
AL,6,7,8,9,3
CSKP,11,0,2,6,7                         * DEFINE CRACK-TIP COORDINATE SYSTEM
CSYS,0
C********************** GENERATE THE FINITE ELEMENT MODEL ************************
ET,1,2,,,2              * STIF2 WITH PLANE STRAIN OPTION
```

```
KSCON,2,A/20,1,6          * DEFINE LOCAL CRACK-TIP MESH
KDVS,ALL,W/3              * DEFINE MAXIMUN DIVISION SIZE
KDVS,1,A/3
KDVS,3,A/3
KDVS,2,A/30
LDVA                      * ADJUST DIVISION SIZES
AMESH,ALL
WSORT,Y                   * REORDER MODEL
C********** DEFINE LOADS AND BOUNDARY CONDITIONS **********
ITER,1,0,1
NSEL,Y,0  $D,ALL,UY,0                  * CONSTRAIN THE BOTTOM NODES
NRSEL,X,0  $D,ALL,UX,0                 * PREVENT RIGID BODY MOTION
NALL
PSF,0,2,L,-SIG                          * APPLY TENSILE STRESS = 100 PSI
LWRITE
AFWRITE
FINISH
/INPUT,27
FINISH
/POST1
C********** COMPUTE KI, KII, AND KIII **********
SET
CSYS,11                    * SET UP LOCAL CRACK-TIP COORDINATES(DEFINED IN PREP7)
LPATH,18,478,477,43,44     * DEFINE PATH FOR KCALC COMMAND
KCALC,,,3,1                * COMPUTE KI,KII,KIII (PLANE STRAIN, FULL-CRACK MODEL)
*GET,KI,KCAL,KI            * GET KI, KII, AND KIII AS PARAMETERS
*GET,KII,KCAL,KII
*GET,KIII,KCAL,KIII
KI=KI/(SIG*(A**.5))        * NORMALIZE K BY DIVIDING BY SIG*SQRT(A)
KII=KII/(SIG*(A**.5))
KIII=KIII/(SIG*(A**.5))
*STAT                      * VIEW RESULTS
FINISH
```

8.5 Compact Tension Specimen (Fine Model) J – Integral Path Evaluations

```
/PREP7
/TITLE, EX. 3-1: COMPACT TENSION SPECIMEN
C********** DEFINE PARAMETERS **********
A=.0466              * DEFINE CRACK LENGTH (M)
B=.003               * DEFINE THICKNESS (M)
W=.100               * DEFINE WIDTH TO LOAD POINT (M)
W1=.125              * DEFINE TOTAL WIDTH (M)
H=.060               * DEFINE HALF HEIGHT (M)
R=.0125              * DEFINE RADIUS OF LOAD CIRCLE (M)
E=.0275              * DEFINE PIN HEIGHT (M)
S=.003               * DEFINE HALF WIDTH OF NOTCH (M)
D1=.080              * DEFINE DEPTH OF NOTCH (M)
D2=.075              * DEFINE DEPTH OF NOTCH (M)
C********** DEFINE MATERIAL PROPERTIES **********
MP,EX,1,211E9                          * YOUNGS'S MODULUS = 211GPA
MP,NUXY,1,.33                          * POISSON'S RATIO = 0.33
```

```
KNL,1
NL,1,13,18,.0023148,.010,.040,.14,.36          * STRESS-STRAIN DATA
NL,1,19,0,488.43E6,488.43E6,660E6,830E6,980E6  * MULTILINEAR ISOTROPIC (K13=18)
C****************************** DEFINE SOLID MODEL ******************************
K,1,A  $K,2,W  $K,3,W,H  $K,4,,H  $K,5,(W-W1),H      * KEYPOINTS
K,6,(W-W1),S  $K,7,,S  $K,8,(W-D1),S  $K,9,(W-D2)
K,10,,E  $K,11,,E,E
CIRCLE,10,R,11,4,,8                                  * HOLE
L,1,2  $RP8,1,1  $L,9,1  $L,4,12  $L,16,7            * LINES
KPSEL,X,-1E-6,1  $LSKP,1  $AL,ALL                    * AREAS
KPSEL,X,-1,1E-6  $LSKP,1  $AL,ALL  $KPALL  $LSALL
C*********************** GENERATE THE FINITE ELEMENT MODEL ***********************
ET,1,2,,,3              * STIF2 WITH PLANE STRESS OPTION (THICHNESS INPUT)
R,1,B                   * THICKNESS = B = .003M
ELSIZE,A/4              * MAXIMUM ELEMENT SIZE
KSCON,1,A/16,0,9        * DEFINE LOCAL CRACK-TIP MESH
AMESH,ALL
WSORT,X                 * REORDER MODEL
C************************ APPLY LOAD AND BOUNDARY CONDITIONS ************************
ITER,-10,0,10
KSE,1                                  * SAVE STRAIN ENERGY FOR POST1
NSEL,Y  $NRSEL,X,A,W  $D,ALL,UY,0      * IMPOSE SYMMETRY AT Y=0
NRSEL,X,A  $D,ALL,UX,0  $NALL          * RIGID BODY CONSTRAINT AT CRACK TIP
*CREATE,LOAD                          * CREATE LOAD MACRO
KF,12,FY,ARG1                         * APPLY PIN LOAD
LWRITE
*END
*USE,LOAD,0.0
RP35,,500                              * LOAD TO 17KN IN STEPS OF .5KN
AFWRITE
FINISH
/INPUT,27
FINISH
/POST1
C************************ COMPUTE J-INTEGRAL VALUES IN POST1 ************************
STRESS,SENE                * STORE STRAIN ENERGY PER ELEMENT
STRESS,VOLU                * STORE VOLUME PER ELEMENT
LOCAL,11,0,A               * DEFINE LOCAL CRACK-TIP COORDINATES
STEP=2                     * START AT LOAD STEP 2
:TOP
SET,STEP,10
CSYS,0
NSEL,X  $NRSEL,Y,S         * SELECT CLIP GAUGE NODE
NSORT,UY
*GET,DISP,MIN              * GET CLIP GAUGE DISPLACEMENT
DISP=(2000*DISP)           * DOUBLE DUE TO SYMMETRY AND CONVERT TO MM
NALL
NSEL,X,0   $NRSEL,Y,(E+R)  * SELECT LOAD POINT NODE
FSUM
*GET,FORC,FSUM,FY          * TRACK APPLIED FORCE (KN)
FORC=-(FORC/1000)
NALL
CSYS,11
```

```
JAVG=0
*USE,JIN1,47,272,279,73     * CALCULATE J FOR FIRST PATH
J1=JINT/1000                * SAVE AS J1 AND CONVERT TO N-MM/MM*MM
JAVG=JAVG+J1                * SUM FOR AVERAGE J-INTEGRAL
*USE,JIN1,49,294,304,71     * SECOND PATH
J2=JINT/1000
JAVG=JAVG+J2
*USE,JIN1,51,361,373,69     * THIRD PATH
J3=JINT/1000
JAVG=JAVG+J3
*USE,JIN1,55,453,455,68     * FOURTH PATH
J4=JINT/1000
JAVG=JAVG+J4
*USE,JIN1,57,472,180,66     * FIFTH PATH
J5=JINT/1000
JAVG=JAVG+J5
*USE,JIN1,59,479,460,65     * SIXTH PATH
J6=JINT/1000
JAVG=JAVG+J6
*USE,JIN1,61,414,423,81     * SEVENTH PATH
J7=JINT/1000
JAVG=JAVG+J7
JAVG=(JAVG/7)               * CALCULATE AVERAVE J-INTEGRAL (N-MM/MM*MM)
*STAT                       * VIEW RESULTS
STEP=(STEP+3)
*IF,STEP,LE,35,:TOP         * REPEAT FOR EVERY THIRD LOAD STEP
FINISH
/POST26
C******** VIEW LOAD VS CLIP GAUGE DISPLACEMENTS AND CALCULATE WORK DONE ********
DISP,2,78,UY                * RETRIEVE CLIP GAUGE DISPLACEMENTS (NODE 78)
DISP,3,8,UY                 * RETRIEVE LOAD POINT DISPLACEMENTS (NODE 8)
FILL,4,1,35,1,0,500         * DEFINE PIN LOAD DATA (N)
ADD,2,2,,,CG-,DISP,,2000    * DOUBLE CLIP GAUGE DISP. (SYM) AND CONVERT TO MM
ADD,3,3,,,LP-,DISP,,2000    * DOUBLE LOAD POINT DISP. (SYM) AND CONVERT TO MM
ADD,4,4,,,FORC,E            * TAG LABEL TO PIN LOAD (N)
INT1,5,4,3,,WORK            * INTEGRATE FOR WORK DONE BY PIN LOADS (N-MM)
ADD,4,4,,,FORC,E,,.001      * CONVERT LOAD TO KN UNITS
PRVAR,2,3,4,5               * SHOW RESULTS
XVAR,2
/GRAPH,LABX,DISP
/GRAPH,LABY,FORC
PLVAR,4                     * VIEW LOAD (KN) VS CLIP GAUGE DISPLACEMENTS (MM)
FINISH
```

8.6 Compact Tension Specimen (Coarse Model)

```
/PREP7
/TITLE, EX. 3-2: COMPACT TENSION SPECIMEN
C****************************** DEFINE PARAMETERS ***********************************
A=.0466                 * DEFINE CRACK LENGTH (M)
B=.003                  * DEFINE THICKNESS (M)
W=.100                  * DEFINE WIDTH TO LOAD POINT (M)
```

```
W1=.125              * DEFINE TOTAL WIDTH (M)
H=.060               * DEFINE HALF HEIGHT (M)
R=.0125              * DEFINE RADIUS OF LOAD CIRCLE (M)
E=.0275              * DEFINE PIN HEIGHT (M)
S=.003               * DEFINE HALF WIDTH OF NOTCH (M)
D1=.080              * DEFINE DEPTH OF NOTCH (M)
D2=.075              * DEFINE DEPTH OF NOTCH (M)
C************************ DEFINE MATERIAL PROPERTIES ****************************
MP,EX,1,211E9                                    * YOUNGS'S MODULUS = 211GPA
MP,NUXY,1,.33                                    * POISSON'S RATIO = 0.33
KNL,1
NL,1,13,18,.0023148,.010,.040,.14,.36            * STRESS-STRAIN DATA
NL,1,19,0,488.43E6,488.43E6,660E6,830E6,980E6    * MULTILINEAR ISOTROPIC (K13=18)
C****************************** DEFINE SOLID MODEL ********************************
K,1,A  $K,2,W  $K,3,W,H  $K,4,,H  $K,5,(W-W1),H      * KEYPOINTS
K,6,(W-W1),S  $K,7,,S  $K,8,(W-D1),S  $K,9,(W-D2)
K,10,,E  $K,11,,E,E
CIRCLE,10,R,11,4,,6                                  * HOLE
L,1,2  $RP8,1,1  $L,9,1  $L,4,12  $L,15,7            * LINES
KPSEL,X,-1E-6,1  $LSKP,1  $AL,ALL                    * AREAS
KPSEL,X,-1,1E-6  $LSKP,1  $AL,ALL  $KPALL  $LSALL
C************************ GENERATE THE FINITE ELEMENT MODEL ************************
ET,1,2,,,3              * STIF2 WITH PLANE STRESS OPTION (THICHNESS INPUT)
R,1,B                   * THICKNESS = B = .003M
ELSIZE,A/3              * MAXIMUM ELEMENT SIZE
KSCON,1,A/10,0,4        * DEFINE LOCAL CRACK-TIP MESH
AMESH,ALL
WSORT,X                 * REORDER MODEL
C************************ APPLY LOAD AND BOUNDARY CONDITIONS ***********************
ITER,-10,0,10
KSE,1                                  * SAVE STRAIN ENERGY FOR POST1
NSEL,Y  $NRSEL,X,A,W  $D,ALL,UY,0      * IMPOSE SYMMETRY AT Y=0
NRSEL,X,A  $D,ALL,UX,0  $NALL          * RIGID BODY CONSTRAINT AT CRACK TIP
*CREATE,LOAD                          * CREATE LOAD MACRO
KF,12,FY,ARG1                         * APPLY PIN LOAD
LWRITE
*END
*USE,LOAD,0.0
RP35,,500                              * LOAD TO 17KN IN STEPS OF .5KN
AFWRITE
FINISH
/INPUT,27
FINISH
/POST1
C************************ COMPUTE J-INTEGRAL VALUES IN POST1 ***********************
STRESS,SENE               * STORE STRAIN ENERGY PER ELEMENT
STRESS,VOLU               * STORE VOLUME PER ELEMENT
LOCAL,11,0,A              * DEFINE LOCAL CRACK-TIP COORDINATES
STEP=2                    * START AT LOAD STEP 2
:TOP
SET,STEP,10
CSYS,0
NSEL,X  $NRSEL,Y,S        * SELECT CLIP GAUGE NODE
```

```
NSORT,UY
*GET,DISP,MIN                * GET CLIP GAUGE DISPLACEMENT
DISP=(2000*DISP)             * DOUBLE DUE TO SYMMETRY AND CONVERT TO MM
NALL
NSEL,X,0    $NRSEL,Y,(E+R)   * SELECT LOAD POINT NODE
FSUM
*GET,FORC,FSUM,FY            * TRACK APPLIED FORCE (KN)
FORC=-(FORC/1000)
NALL
CSYS,11
JAVG=0
*USE,JIN1,37,162,166,55      * CALCULATE J FOR FIRST PATH
J1=JINT/1000                 * SAVE AS J1 AND CONVERT TO N-MM/MM*MM
JAVG=JAVG+J1                 * SUM FOR AVERAGE J-INTEGRAL
*USE,JIN1,39,170,178,53      * SECOND PATH
J2=JINT/1000
JAVG=JAVG+J2
*USE,JIN1,41,235,242,51      * THIRD PATH
J3=JINT/1000
JAVG=JAVG+J3
*USE,JIN1,43,239,230,50      * FOURTH PATH
J4=JINT/1000
JAVG=JAVG+J4
*USE,JIN1,45,223,92,49       * FIFTH PATH
J5=JINT/1000
JAVG=JAVG+J5
*USE,JIN1,47,82,75,63        * SIXTH PATH
J6=JINT/1000
JAVG=JAVG+J6
JAVG=(JAVG/6)                * CALCULATE AVERAVE J-INTEGRAL (N-MM/MM*MM)
*STAT                        * VIEW RESULTS
STEP=(STEP+3)
*IF,STEP,LE,35,:TOP          * REPEAT FOR EVERY THIRD LOAD STEP
FINISH
/POST26
C********* VIEW LOAD VS CLIP GAUGE DISPLACEMENTS AND CALCULATE WORK DONE *********
DISP,2,60,UY                * RETRIEVE CLIP GAUGE DISPLACEMENTS (NODE 60)
DISP,3,6,UY                 * RETRIEVE LOAD POINT DISPLACEMENTS (NODE 6)
/PREP7
/TITLE, EX. 3-2: COMPACT TENSION SPECIMEN
C****************************** DEFINE PARAMETERS ***********************************
A=.0466              * DEFINE CRACK LENGTH (M)
B=.003               * DEFINE THICKNESS (M)
W=.100               * DEFINE WIDTH TO LOAD POINT (M)
W1=.125              * DEFINE TOTAL WIDTH (M)
H=.060               * DEFINE HALF HEIGHT (M)
R=.0125              * DEFINE RADIUS OF LOAD CIRCLE (M)
E=.0275              * DEFINE PIN HEIGHT (M)
S=.003               * DEFINE HALF WIDTH OF NOTCH (M)
D1=.080              * DEFINE DEPTH OF NOTCH (M)
D2=.075              * DEFINE DEPTH OF NOTCH (M)
C************************** DEFINE MATERIAL PROPERTIES ****************************
MP,EX,1,211E9                                  * YOUNGS'S MODULUS = 211GPA
```

```
MP,NUXY,1,.33                                    * POISSON'S RATIO = 0.33
KNL,1
NL,1,13,18,.0023148,.010,.040,.14,.36            * STRESS-STRAIN DATA
NL,1,19,0,488.43E6,488.43E6,660E6,830E6,980E6  * MULTILINEAR ISOTROPIC (K13=18)
C******************************* DEFINE SOLID MODEL ******************************
K,1,A  $K,2,W  $K,3,W,H  $K,4,,H  $K,5,(W-W1),H      * KEYPOINTS
K,6,(W-W1),S  $K,7,,S  $K,8,(W-D1),S  $K,9,(W-D2)
K,10,,E  $K,11,,E,E
CIRCLE,10,R,11,4,,6                                  * HOLE
L,1,2  $RP8,1,1  $L,9,1  $L,4,12  $L,15,7            * LINES
KPSEL,X,-1E-6,1  $LSKP,1  $AL,ALL                    * AREAS
KPSEL,X,-1,1E-6  $LSKP,1  $AL,ALL  $KPALL  $LSALL
C*********************** GENERATE THE FINITE ELEMENT MODEL ***********************
ET,1,2,,,3              * STIF2 WITH PLANE STRESS OPTION (THICHNESS INPUT)
R,1,B                   * THICKNESS = B = .003M
ELSIZE,A/3              * MAXIMUM ELEMENT SIZE
KSCON,1,A/10,0,4        * DEFINE LOCAL CRACK-TIP MESH
AMESH,ALL
WSORT,X                 * REORDER MODEL
C************************ APPLY LOAD AND BOUNDARY CONDITIONS ***********************
ITER,-10,0,10
KSE,1                                  * SAVE STRAIN ENERGY FOR POST1
NSEL,Y  $NRSEL,X,A,W  $D,ALL,UY,0      * IMPOSE SYMMETRY AT Y=0
NRSEL,X,A  $D,ALL,UX,0  $NALL          * RIGID BODY CONSTRAINT AT CRACK TIP
*CREATE,LOAD                           * CREATE LOAD MACRO
KF,12,FY,ARG1                          * APPLY PIN LOAD
LWRITE
*END
*USE,LOAD,0.0
RP35,,500                              * LOAD TO 17KN IN STEPS OF .5KN
AFWRITE
FINISH
/INPUT,27
FINISH
/POST1
C************************ COMPUTE J-INTEGRAL VALUES IN POST1 ***********************
STRESS,SENE                * STORE STRAIN ENERGY PER ELEMENT
STRESS,VOLU                * STORE VOLUME PER ELEMENT
LOCAL,11,0,A               * DEFINE LOCAL CRACK-TIP COORDINATES
STEP=2                     * START AT LOAD STEP 2
:TOP
SET,STEP,10
CSYS,0
NSEL,X  $NRSEL,Y,S         * SELECT CLIP GAUGE NODE
NSORT,UY
*GET,DISP,MIN              * GET CLIP GAUGE DISPLACEMENT
DISP=(2000*DISP)           * DOUBLE DUE TO SYMMETRY AND CONVERT TO MM
NALL
NSEL,X,0   $NRSEL,Y,(E+R)  * SELECT LOAD POINT NODE
FSUM
*GET,FORC,FSUM,FY          * TRACK APPLIED FORCE (KN)
FORC=-(FORC/1000)
NALL
```

```
CSYS,11
JAVG=0
*USE,JIN1,37,162,166,55     * CALCULATE J FOR FIRST PATH
J1=JINT/1000                * SAVE AS J1 AND CONVERT TO N-MM/MM*MM
JAVG=JAVG+J1                * SUM FOR AVERAGE J-INTEGRAL
*USE,JIN1,39,170,178,53     * SECOND PATH
J2=JINT/1000
JAVG=JAVG+J2
*USE,JIN1,41,235,242,51     * THIRD PATH
J3=JINT/1000
JAVG=JAVG+J3
*USE,JIN1,43,239,230,50     * FOURTH PATH
J4=JINT/1000
JAVG=JAVG+J4
*USE,JIN1,45,223,92,49      * FIFTH PATH
J5=JINT/1000
JAVG=JAVG+J5
*USE,JIN1,47,82,75,63       * SIXTH PATH
J6=JINT/1000
JAVG=JAVG+J6
JAVG=(JAVG/6)               * CALCULATE AVERAVE J-INTEGRAL (N-MM/MM*MM)
*STAT                       * VIEW RESULTS
STEP=(STEP+3)
*IF,STEP,LE,35,.TOP         * REPEAT FOR EVERY THIRD LOAD STEP
FINISH
/POST26
C******** VIEW LOAD VS CLIP GAUGE DISPLACEMENTS AND CALCULATE WORK DONE ********
DISP,2,60,UY                * RETRIEVE CLIP GAUGE DISPLACEMENTS (NODE 60)
DISP,3,6,UY                 * RETRIEVE LOAD POINT DISPLACEMENTS (NODE 6)
FILL,4,1,35,1,0,500         * DEFINE PIN LOAD DATA (N)
ADD,2,2,,,CG-,DISP,,2000    * DOUBLE CLIP GAUGE DISP. (SYM) AND CONVERT TO MM
ADD,3,3,,,LP-,DISP,,2000    * DOUBLE LOAD POINT DISP. (SYM) AND CONVERT TO MM
ADD,4,4,,,FORC,E            * TAG LABEL TO PIN LOAD (N)
INT1,5,4,3,,WORK            * INTEGRATE FOR WORK DONE BY PIN LOADS (N-MM)
ADD,4,4,,,FORC,E,,.001      * CONVERT LOAD TO KN UNITS
PRVAR,2,3,4,5               * SHOW RESULTS
XVAR,2
/GRAPH,LABX,DISP
/GRAPH,LABY,FORC
PLVAR,4                     * VIEW LOAD (KN) VS CLIP GAUGE DISPLACEMENTS (MM)
FINISH
FILL,4,1,35,1,0,500         * DEFINE PIN LOAD DATA (N)
ADD,2,2,,,CG-,DISP,,2000    * DOUBLE CLIP GAUGE DISP. (SYM) AND CONVERT TO MM
ADD,3,3,,,LP-,DISP,,2000    * DOUBLE LOAD POINT DISP. (SYM) AND CONVERT TO MM
ADD,4,4,,,FORC,E            * TAG LABEL TO PIN LOAD (N)
INT1,5,4,3,,WORK            * INTEGRATE FOR WORK DONE BY PIN LOADS (N-MM)
ADD,4,4,,,FORC,E,,.001      * CONVERT LOAD TO KN UNITS
PRVAR,2,3,4,5               * SHOW RESULTS
XVAR,2
/GRAPH,LABX,DISP
/GRAPH,LABY,FORC
PLVAR,4                     * VIEW LOAD (KN) VS CLIP GAUGE DISPLACEMENTS (MM)
FINISH
```

8.7 Compact Tension Specimen (Coarse Model) Using POST26

```
/PREP7
/TITLE, EX. 3-3: COMPACT TENSION SPECIMEN
C*************************** DEFINE PARAMETERS ***********************************
A=.0466              * DEFINE CRACK LENGTH (M)
B=.003               * DEFINE THICKNESS (M)
W=.100               * DEFINE WIDTH TO LOAD POINT (M)
W1=.125              * DEFINE TOTAL WIDTH (M)
H=.060               * DEFINE HALF HEIGHT (M)
R=.0125              * DEFINE RADIUS OF LOAD CIRCLE (M)
E=.0275              * DEFINE PIN HEIGHT (M)
S=.003               * DEFINE HALF WIDTH OF NOTCH (M)
D1=.080              * DEFINE DEPTH OF NOTCH (M)
D2=.075              * DEFINE DEPTH OF NOTCH (M)
C************************* DEFINE MATERIAL PROPERTIES ******************************
MP,EX,1,211E9                                  * YOUNGS'S MODULUS = 211GPA
MP,NUXY,1,.33                                  * POISSON'S RATIO = 0.33
KNL,1
NL,1,13,18,.0023148,.010,.040,.14,.36          * STRESS-STRAIN DATA
NL,1,19,0,488.43E6,488.43E6,660E6,830E6,980E6  * MULTILINEAR ISOTROPIC (K13=18)
C******************************* DEFINE SOLID MODEL ********************************
K,1,A  $K,2,W  $K,3,W,H  $K,4,,H  $K,5,(W-W1),H      * KEYPOINTS
K,6,(W-W1),S  $K,7,,S  $K,8,(W-D1),S  $K,9,(W-D2)
K,10,,E  $K,11,,E,E
CIRCLE,10,R,11,4,,6                                  * HOLE
L,1,2  $RP8,1,1  $L,9,1  $L,4,12  $L,15,7            * LINES
KPSEL,X,-1E-6,1  $LSKP,1  $AL,ALL                    * AREAS
KPSEL,X,-1,1E-6  $LSKP,1  $AL,ALL  $KPALL  $LSALL
C************************ GENERATE THE FINITE ELEMENT MODEL ************************
ET,1,2,,,3              * STIF2 WITH PLANE STRESS OPTION (THICHNESS INPUT)
R,1,B                   * THICKNESS = B = .003M
ELSIZE,A/3              * MAXIMUM ELEMENT SIZE
KSCON,1,A/10,0,4        * DEFINE LOCAL CRACK-TIP MESH
AMESH,ALL
WSORT,X                 * REORDER MODEL
C************************ APPLY LOAD AND BOUNDARY CONDITIONS ***********************
ITER,-10,0,10
KSE,1                                  * SAVE STRAIN ENERGY FOR POST1
NSEL,Y  $NRSEL,X,A,W  $D,ALL,UY,0      * IMPOSE SYMMETRY AT Y=0
NRSEL,X,A  $D,ALL,UX,0  $NALL          * RIGID BODY CONSTRAINT AT CRACK TIP
*CREATE,LOAD                           * CREATE LOAD MACRO
KF,12,FY,ARG1                          * APPLY PIN LOAD
LWRITE
*END
*USE,LOAD,0.0
RP35,,500                              * LOAD TO 17KN IN STEPS OF .5KN
AFWRITE
FINISH
C********************* FIRST ANALYSIS (NONLINEAR, A=.0466M) ************************
/INPUT,27
FINISH
/COPY,12,10,0                          * SAVE RESULTS AS FILE10
```

```
/PREP7
C********** SET UP FIRST ELASTIC MODEL (A1=.0466) **********
RESUME
A1=A
KNL,0
ITER,1,0,1
NSEL,Y  $NRSEL,X,A,W  $D,ALL,UY,0       * IMPOSE SYMMETRY AT Y=0
NRSEL,X,A  $D,ALL,UX,0  $NALL           * RIGID BODY CONSTRAINT AT CRACK TIP
KD,12,UY,1.0                            * IMPOSE PIN DISPLACEMENT (UNITY)
LWRITE
AFWRITE
FINISH
C********** SECOND ANALYSIS (LINEAR, ONE ITERATION, A=A1) **********
/INPUT,27
FINISH
/POST1
C********** IN POST1 GET PIN LOAD P1 FOR A = A1 **********
SET,1,1
NSEL,X   $NRSEL,Y,(E+R)       * SELECT PIN NODE
FSUM
*GET,P1,FSUM,FY               * GET PIN LOAD P1 (N)
P1=P1*2
FINISH
/PREP7
C********** SET UP SECOND ELASTIC MODEL (A=A2=.0466+.000466M) **********
RESUME,,,,1
A2=(A+(A/100))
FACT=(A2/A1)                        * FACTOR FOR SCALING = 1.01
MODMESH,NOCHECK                     * ALLOW NODE MODIFICATIONS
NSEL,X,A  $NRSEL,Y                  * SELECT NODES NEAR THE CRACK TIP
ENODE  $NELEM
ENODE  $NELEM
NSCALE,0,ALL,,,FACT                 * EXTEND CRACK ONE PERCENT
NALL  $EALL
KNL,0                               * LINEAR ELASTIC ANALYSIS
ITER,1,0,1
NSEL,Y  $NRSEL,X,A2,W  $D,ALL,UY,0      * IMPOSE SYMMETRY AT Y=0
NRSEL,X,A2  $D,ALL,UX,0  $NALL          * RIGID BODY CONSTRAINT AT CRACK TIP
KD,12,UY,1.0                            * IMPOSE PIN DISPLACEMENT (UNITY)
LWRITE
AFWRITE
FINISH
C********** THIRD ANALYSIS (LINEAR, ONE ITERATION, A=A2) **********
/INPUT,27
FINISH
/POST1
C********** IN POST1 GET PIN LOAD P2 FOR A = A2 **********
SET,1,1
NSEL,X,0  $NRSEL,Y,(E+R)  * SELECT PIN NODE
FSUM
*GET,P2,FSUM,FY           * GET PIN LOAD P2 (N)
P2=P2*2
DA=((A2-A1)*1000)         * CHANGE IN CRACK LENGTH (MM)
```

```
DPDA=((P2-P1)/DA)          * CALC. CHANGE IN FORCE W.R.T. CHANGE IN CRACK LENGTH
FINISH                     * UNITS ARE N/MM
/POST26
C************ USE POST26 TO ESTIMATE J FROM LOAD VS. LOAD POINT WORK ***********
FILE,10                    * FIRST MODEL (FULL NONLINEAR ANALYSIS)
DISP,2,6,UY                * RETRIEVE LOAD POINT DISPLACEMENTS, M (NODE 6)
DISP,6,60,UY               * RETRIEVE CLIP GAUGE DISPLACEMENTS, M (NODE 60)
FILL,3,1,35,1,0,500        * DEFINE PIN LOAD (N)
ADD,3,3,,,FORC,E           * TAG LABEL TO PIN LOAD
ADD,2,2,,,DISP,-LP,,2000   * DOUBLE LOAD POINT DISP. (SYM) AND CONVERT TO MM
ADD,6,6,,,DISP,-CG,,2000   * DOUBLE CLIP GAUGE DISP. (SYM) AND CONVERT TO MM
INT1,4,3,2,,WORK,          * INTEGRATE FOR WORK DONE BY PIN LOADS (N-MM)
CON=(DPDA/(B*P1))          * CALC. CONSTANT (DP/DA)/B*P  (1/MM*MM)
CON=CON/1000
ADD,5,4,,,JINT,,,CON       * ESTIMATE J (N-MM/MM**2)
PRVAR,2,3,4,5,6            * LIST RESULTS
*STAT
FINISH
```

8.8 3 – D Semicircular Surface Flaw : Elastic Analysis

```
/PREP7
/TITLE, EX. 4: 3-D SEMI-CIRCULAR SURFACE FLAW
C****************************** FULL MODEL GENERATION ******************************
C******************************** DEFINE PARAMETERS ********************************
A=1.0          * DEFINE CRACK LENGTH
R=A            * DEFINE CRACK RADIUS
B=(5*A)        * DEFINE PLATE LENGTH
C=A            * DEFINE CRACK WIDTH
T=(2.5*A)      * DEFINE PLATE WIDTH
H=(5*A)        * DEFINE PLATE HEIGHT
SIG=100        * DEFINE APPLIED LOAD
E=30E6         * YOUNG'S MODULUS
NU=0.3         * POISSON'S RATIO
C****************************** BUILD THE FULL SOLID MODEL ***************************
K,1,0        $K,2,B       $K,3,B,T     $K,4,0,T        * KEYPOINTS
K,5,0,0,H    $K,6,B,0,H   $K,7,B,T,H   $K,8,0,T,H
K,9,R,0,0    $K,10,0,R,0
LARC,9,10,1,R,1                                       * CIRCULAR LINE (CRACK)
L,9,2     $L,2,3     $L,3,4     $L,4,10              * REMAINING LINES
L,1,9     $L,1,10    $L,1,5     $L,5,6
L,6,2     $L,5,8     $L,8,4
AL,1,2,3,4,5    $A,1,9,10     $AL,6,2,10,9,8         * AREAS
A,2,3,7,6       $A,3,4,8,7    $AL,5,7,8,11,12
A,6,7,8,5
VA,ALL                                                * VOLUME
C********************* GENERATE THE FULL FINITE ELEMENT MODEL **********************
ET,1,92                        * 10-NODE TETRAHEDRONS
MP,EX,1,E                      * MATERIAL PROPERTIES
MP,NUXY,1,NU
KDVS,ALL,T                     * MAX. ELEMENT SIZE = T = 2.5*A
KDVS,9,A/4                     * ELEMENT SIZE = A/4 AT CRACK FRONT
```

```
KDVS,10,A/4
KDVS,1,A/2
KDVS,2,T/1.5
KDVS,4,A/2
LDVA
VMESH,1
CSYS,2
WSORT,X                          * REORDER MODEL
CSYS,0
C********************** APPLY LOADS AND BOUNDARY CONDITIONS ************************
ITER,1,0,1
ASBC,1                * APPLY SYMMETRY BOUNDARY CONDITIONS AT Z=0
ASBC,6                * APPLY SYMMETRY BOUNDARY CONDITIONS AT X=0
KD,1,UY,0             * PREVENT RIGID BODY TRANSLATION AT X=Y=Z=0
APSF,7,-SIG           * APPLY UNIFORM TENSILE STRESS AT Z=H
AFWRITE
FINISH
C************************ ANSLYSIS OF THE FULL MODEL ********************************
/INPUT,27
FINISH
/COPY,12,10,0         * SAVE FILE12 AS FILE10 FOR AUX1
/PREP7
C************************* GENERATE THE SUBMODEL ************************************
MP,EX,1,E  $MP,NUXY,1,NU      * MATERIAL PROPERTIES
C********************** DEFINE SIZE OF EACH ELEMENT LAYER ***************************
R1=A/8             * SIZE OF SINGULAR ELEMENTS
R2=A/3             * SECOND LAYER SIZE
R3=A*(2/3)         * THIRD LAYER SIZE
C*************************** DEFINE SUBMODEL NODES **********************************
LOCAL,11,1,A,,,,,90
N,1               $RP7,1                        * CRACK-FRONT NODES
N,8,R1/4          $NGEN,7,1,8,,,,30             * NODES AT 1/4 POINTS
N,15,R1           $NGEN,7,1,15,,,,30            * NODES AT R=R1
CSYS,0
FILL,15,16,1,22
RP6,1,1,,1
CSYS,11
N,35,R2           $NGEN,7,1,35,,,,30            * NODES AT R=R2
FILL,15,35,1,28
RP7,1,1,,1
CSYS,0
FILL,35,36,1,42
RP6,1,1,,1
CSYS,11
N,1255,R3         $NGEN,7,1,1255,,,,30          * NODES AT R=R3
FILL,35,1255,1,48
RP7,1,1,,1
CSYS,0
FILL,1255,1256,1,1262
RP6,1,1,,1
CSYS,1
NGEN,11,100,ALL,,,,9
CSYS,0
```

```
FILL,1,101,1,61
RP7,1,1,,1
FILL,15,115,1,68
RP7,1,1,,1
FILL,35,135,1,75
RP7,1,1,,1
FILL,1255,1355,1,1271
RP7,1,1,,1
CSYS,1
NGEN,10,100,61,81,1,,9
NGEN,10,100,1271,1277,1,,9
CSYS,0
C************************ DEFINE SUBMODEL ELEMENTS ******************************
ET,1,95                                          * 20-NODE SOLID ELEMENTS
E,115,116,101,101,15,16,1,1                      * SINGULAR WEDGES
EMORE,122,109,101,108,22,9,1,8
EMORE,68,69,61,61
EGEN,6,1,1
E,135,136,116,115,35,36,16,15                    * REGULAR ELEMENTS
EMORE,142,129,122,128,42,29,22,28
EMORE,75,76,69,68
EGEN,6,1,-1
E,1355,1356,136,135,1255,1256,36,35              * REGULAR ELEMENTS
EMORE,1362,149,142,148,1262,49,42,48
EMORE,1271,1272,76,75
EGEN,6,1,-1
EGEN,10,100,-18                                  * COMPLETE THE MODEL
C*********************** ADDITIONAL NODE MODIFICATIONS ***************************
NUMM,NODE                          * MERGE COINCIDENT NODES
NSEL,NODE,1200,5000                * SELECT CUT-BOUNDARY NODES FOR AUX1
NWRITE                             * PUT SUBMODEL NODES ON FILE15
NALL
FINISH
/AUX1
C******** IN AUX1, CONVERT FULL-MODEL DISPLACEMENTS TO SUBMODEL SURFACE *********
CBDSP,10,15,39   * 10 IS FULL MODEL RESULTS, 15 IS SUBMODEL NODES, 39 IS D CMDS
FINISH
/PREP7
C************* APPLY BOUNDARY CONDITIONS TO SUBMODEL AND ANALYZE IT *************
RESUME
ITER,1,0,1
/INPUT,39                            * IMPOSE AUX1 CUT-BOUNDARY DISPLACEMENTS
CSYS,1  $NSEL,Z                      * IMPOSE UZ=0 ON SYMMETRY PLANE Z=0
NRSEL,X,(.99*A),10*A
D,ALL,UZ
NALL  $CSYS,0  $NSEL,X,0             * IMPOSE UX=0 ON SYMMETRY PLANE X=0
D,ALL,UX
NALL
AFWRITE
FINISH
/INPUT,27
FINISH
/POST1
```

```
C************** IN POST1, COMPUTE KI ALONG THE SUBMODEL CRACK FRONT *************
CON=SIG*(A**.5)          * CONSTANT = SIG*SQRT(A)
SET
CS,11,0,1,15,17          * AT 0 DEGREES (FREE SURFACE) SET UP LOCAL SYSTEM
LPATH,1,14,21                * DEFINE EXTRAPOLATION NODES FOR THE KCALC COMMAND
KCALC,1                      * COMPUTE KI (ASSUME PLANE STRESS AT FREE SURFACE)
*GET,KI0,KCALC,KI            * GET KI AT 0 DEGREES AS A PARAMETER KI0=KI
KI0=KI0/CON                  * MAKE KI DIMENSIONLESS
CS,11,0,101,115,117      * AT 9 DEGREES
LPATH,101,114,121
KCALC
*GET,KI9,KCALC,KI
KI9=KI9/CON
CS,11,0,201,215,217      * AT 18 DEGREES
LPATH,201,214,221
KCALC
*GET,KI18,KCALC,KI
KI18=KI18/CON
CS,11,0,301,315,317      * AT 27 DEGREES
LPATH,301,314,321
KCALC
*GET,KI27,KCALC,KI
KI27=KI27/CON
CS,11,0,401,415,417      * AT 36 DEGREES
LPATH,401,414,421
KCALC
*GET,KI36,KCALC,KI
KI36=KI36/CON
CS,11,0,501,515,517      * AT 45 DEGREES
LPATH,501,514,521
KCALC
*GET,KI45,KCALC,KI
KI45=KI45/CON
CS,11,0,601,615,617      * AT 54 DEGREES
LPATH,601,614,621
KCALC
*GET,KI54,KCALC,KI
KI54=KI54/CON
CS,11,0,701,715,717      * AT 63 DEGREES
LPATH,701,714,721
KCALC
*GET,KI63,KCALC,KI
KI63=KI63/CON
CS,11,0,801,815,817      * AT 72 DEGREES
LPATH,801,814,821
KCALC
*GET,KI72,KCALC,KI
KI72=KI72/CON
CS,11,0,901,915,917      * AT 81 DEGREES
LPATH,901,914,921
KCALC
*GET,KI81,KCALC,KI
KI81=KI81/CON
```

```
CS,11,0,1001,1015,1017     * AT 90 DEGREES (MIDDLE PLANE)
LPATH,1001,1014,1021
KCALC
*GET,KI90,KCALC,KI
KI90=KI90/CON
*STAT                      * LIST RESULTS
FINISH
```

8.9 Macro J – Integral User File for Compact Tension Specimen Analysis

```
C*************************************************************************
C*************************  J-INTEGRAL USER FILE  ************************
C*************************************************************************
C***  THIS POST1 USER FILE CAN BE USED FOR THE EVALUATION OF THE 2-D J-INTEGRAL
C***  GIVEN BY EQUATION (2.3:5) IN THIS TUTORIAL. TO EMPLOY THIS FILE ISSUE
C***  "*USE,JIN1,N1,N2,N3,N4", WHERE N1 TO N4 ARE NODE ARGUMENTS THAT DEFINE
C***  THE J-INTEGRAL PATH. THE FOLLOWING POST1 COMMANDS MUST BE ISSUED BEFORE
C***  THIS FILE IS USED: "STRESS,SENE"; "STRESS,VOLU"; AND "SET". RESTRICTIONS
C***  THAT APPLY TO EQUATION (2.3:5) ARE DISCUSSED IN SECTION 2.3. IN ORDER TO
C***  USE THIS FILE CORRECTLY, THE CRACK MUST LIE IN THE GLOBAL CARTESIAN X-Y
C***  PLANE (CSYS,0), WITH X PARALLEL TO THE CRACK, AND DISPLACEMENT AND STRESS
C***  COMPONENTS MUST BE IN GLOBAL CARTESIAN COORDINATES. OUTPUT IS THE
C***  PARAMETER JINT.
C*************************************************************************
SEXP,W,SENE,VOLU,1,-1        * CALCULATE STRAIN ENERGY DENSITY
LPATH,ARG1,ARG2,ARG3,ARG4    * DEFINE THE PATH FROM FOUR NODAL ARGUMENTS
PDEF,INTR,W,W                * PUT STRAIN ENERGY DENSITY ON THE PATH
PCALC,INTG,J,W,YG            * INTEGRATE ENERGY W.R.T. GLOBAL Y
*GET,JA,PLAST,J              * GET FINAL VALUE OF INTEGRAL FOR 1ST TERM OF J
PDEF,CLEAR                   * CLEAR OLD PATH VARIABLES
PDEF,NORM,NX,NY,NZ           * DEFINE PATH UNIT NORMAL VECTOR
PDEF,INTR,SX,SX              * PUT STRESS SX ON THE PATH
PDEF,INTR,SY,SY              * PUT STRESS SY ON THE PATH
PDEF,INTR,SXY,SXY            * PUT STRESS SXY ON THE PATH
PCALC,MULT,TX,SX,NX          * CALCULATE TRACTION TX
PCALC,MULT,C1,SXY,NY         *   TX = SX*NX + SXY*NY
PCALC,ADD,TX,TX,C1
PCALC,MULT,TY,SXY,NX         * CALCULATE TRACTION TY
PCALC,MULT,C1,SY,NY          *   TY = SXY*NX + SY*NY
PCALC,ADD,TY,TY,C1
*GET,DX,PLAST,S              * DEFINE PATH SHIFT AS 1% OF PATH LENGTH
DX=DX/100
PCALC,ADD,XG,XG,,,,-DX/2     * SHIFT PATH FROM X TO X-DX/2 (GLOBAL X DIR.)
PDEF,INTR,UX1,UX             * DEFINE UX AT X-DX
PDEF,INTR,UY1,UY             * DEFINE UY AT X-DX
PCALC,ADD,XG,XG,,,,DX        * SHIFT PATH FROM X-DX/2 TO X+DX/2
PDEF,INTR,UX2,UX             * DEFINE UX AT X+DX
PDEF,INTR,UY2,UY             * DEFINE UY AT X+DX
PCALC,ADD,XG,XG,,,,-DX/2     * SHIFT PATH BACK TO ORIGINAL POSITION
C=(1/DX)
PCALC,ADD,C1,UX2,UX1,C,-C    * CALCULATE DERIVATIVE DUX/DX
PCALC,ADD,C2,UY2,UY1,C,-C    * CALCULATE DERIVATIVE DUY/DX
```

```
PCALC,MULT,C1,TX,C1          * DEFINE INTEGRAND
PCALC,MULT,C2,TY,C2          *    = TX*DUX/DX + TY*DUY/DX
PCALC,ADD,C1,C1,C2
PCALC,INTG,J,C1,S            * FORM SECOND INTEGRAL (W.R.T. PATH LENGTH S)
*GET,JB,PLAST,J              * GET FINAL VALUE OF INTEGRAL FOR 2ND TERM OF J
JINT=(2*(JA-JB))             * ADD BOTH TERMS AND DOUBLE FOR HALF MODELS
PDEF,CLEAR                   * CLEAR PATH VARIABLES
```

8.10 Macro J – Integral User File for Single – Edge Crack Elasic Analysis

```
C*************************************************************************
C*************************  J-INTEGRAL USER FILE  ************************
C*************************************************************************
C***  THIS POST1 USER FILE CAN BE USED FOR THE EVALUATION OF THE 2-D J-INTEGRAL
C***  GIVEN BY EQUATION (2.3:5) IN THIS TUTORIAL. TO EMPLOY THIS FILE ISSUE
C***  "*USE,JIN2". THE CALCULATION FOR STRAIN ENERGY DENSITY USES NODAL
C***  PRINCIPAL STRESSES AND PARAMETERS E (=YOUNG'S MODULUS) AND NU (=POISSON"S
C***  RATIO). THE POST1 "SET" AND "LPATH" COMMANDS MUST BE ISSUED, AND E AND NU
C***  MUST BE DEFINED BEFORE THIS FILE IS USED. THIS FILE MAY ONLY BE USED WITH
C***  LINEAR ELASTIC ANALYSIS RESULTS. ADDITIONAL RESTRICTIONS THAT APPLY TO
C***  EQUATION (2.3:5) ARE DISCUSSED IN SECTION 2.3. IN ORDER TO USE THIS FILE
C***  CORRECTLY, THE CRACK MUST LIE IN THE GLOBAL CARTESIAN X-Y PLANE (CSYS,0),
C***  WITH X PARALLEL TO THE CRACK, AND DISPLACEMENT AND STRESS COMPONENTS MUST
C***  BE IN GLOBAL CARTESIAN COORDINATES. OUTPUT IS THE PARAMETER JINT.
C*************************************************************************
PDEF,INTR,S1,SIG1            * PUT STRESS SIG1 ON THE PATH
PDEF,INTR,S2,SIG2            * PUT STRESS SIG2 ON THE PATH
PDEF,INTR,S3,SIG3            * PUT STRESS SIG3 ON THE PATH
PCALC,MULT,C11,S1,S1         * CALCULATE STRAIN ENERGY DENSITY FROM PRINCIPAL
PCALC,MULT,C22,S2,S2         *    STRESS COMPONENTS:
PCALC,MULT,C33,S3,S3         *    W = (1/2E)*(S1*S1 + S2*S2 + S3*S3)
PCALC,MULT,C12,S1,S2         *      - (NU/E)*(S1*S2 + S2*S3 + S3*S1)
PCALC,MULT,C23,S2,S3
PCALC,MULT,C31,S3,S1
C=(1/(2*E))
PCALC,ADD,SUM1,C11,C22
PCALC,ADD,SUM1,SUM1,C33,C,C
C=NU/E
PCALC,ADD,SUM2,C12,C23
PCALC,ADD,SUM2,SUM2,C31,C,C
PCALC,ADD,W,SUM1,SUM2,1,-1
PCALC,INTG,J,W,YG            * INTEGRATE ENERGY W.R.T. GLOBAL Y
*GET,JA,PLAST,J              * GET FINAL VALUE OF INTEGRAL FOR 1ST TERM OF J
PDEF,CLEAR                   * CLEAR OLD PATH VARIABLES
PDEF,NORM,NX,NY,NZ           * DEFINE PATH UNIT NORMAL VECTOR
PDEF,INTR,SX,SX              * PUT STRESS SX ON THE PATH
PDEF,INTR,SY,SY              * PUT STRESS SY ON THE PATH
PDEF,INTR,SXY,SXY            * PUT STRESS SXY ON THE PATH
PCALC,MULT,TX,SX,NX          * CALCULATE TRACTION TX
PCALC,MULT,C1,SXY,NY         *   TX = SX*NX + SXY*NY
PCALC,ADD,TX,TX,C1
PCALC,MULT,TY,SXY,NX         * CALCULATE TRACTION TY
```

```
PCALC,MULT,C1,SY,NY          *   TY = SXY*NX + SY*NY
PCALC,ADD,TY,TY,C1
*GET,DX,PLAST,S              * DEFINE PATH SHIFT AS 1% OF PATH LENGTH
DX=DX/100
PCALC,ADD,XG,XG,,,,-DX/2     * SHIFT PATH FROM X TO X-DX/2 (GLOBAL X DIR.)
PDEF,INTR,UX1,UX             * DEFINE UX AT X-DX
PDEF,INTR,UY1,UY             * DEFINE UY AT X-DX
PCALC,ADD,XG,XG,,,,DX        * SHIFT PATH FROM X-DX/2 TO X+DX/2
PDEF,INTR,UX2,UX             * DEFINE UX AT X+DX
PDEF,INTR,UY2,UY             * DEFINE UY AT X+DX
PCALC,ADD,XG,XG,,,,-DX/2     * SHIFT PATH BACK TO ORIGINAL POSITION
C=(1/DX)
PCALC,ADD,C1,UX2,UX1,C,-C    * CALCULATE DERIVATIVE DUX/DX
PCALC,ADD,C2,UY2,UY1,C,-C    * CALCULATE DERIVATIVE DUY/DX
PCALC,MULT,C1,TX,C1          * DEFINE INTEGRAND
PCALC,MULT,C2,TY,C2          *   = TX*DUX/DX + TY*DUY/DX
PCALC,ADD,C1,C1,C2
PCALC,INTG,J,C1,S            * FORM SECOND INTEGRAL (W.R.T. PATH LENGTH S)
*GET,JB,PLAST,J             * GET FINAL VALUE OF INTEGRAL FOR 2ND TERM OF J
JINT=(2*(JA-JB))             * ADD BOTH TERMS AND DOUBLE FOR HALF MODELS
PDEF,CLEAR                   * CLEAR PATH VARIABLES
```

9. Acoustic Pressure Wave Propagation

9.1 Pressure Step Propagation under Water 1 – D Wave

```
C***    1 D Plane Wave
/PREP7
/TITLE, PROPAGATION OF A STEP PRESSURE WAVE UNDER WATER
KAN,4                          * Tansient analysis
KAY,5,2                        * Set first and second time
C***                             derivatives of pressure to zero
ET,1,30,,1                     * Specify element type
K,1                            * Generate F.E. mesh
K,2,1
K,3,1,1
K,4,,1
K,5,,,20
K,6,1,,20
K,7,1,1,20
K,8,,1,20
L,1,2,1
L,2,3,1
L,3,4,1
L,4,1,1
V,1,2,3,4,5,6,7,8
ELSIZE,,20
VMESH,ALL
C***                         Specify Material Properties
MU,1,0                         * Zero damping
VISC,1,1440                    * Speed of sound C (m/s)
DENS,1,1000                    * Density of water (kg/m**3)
C***                         Apply Initial Conditions
NSEL,Z,,10.0                   * Select nodes within Z=0 to 10
NT,ALL,PRES,200.0              * Specify step pressure distribution
NINV
NT,ALL,PRES,100.0
NALL
EALL
ITER,1,,1
KBC,1
LWRITE                         * This is important for initial
C***                             conditions to take effect
TIME,0.01                      * This is the final time
ITER,100,,10                   * Specifies the time step and how
C***                             often the results are written
GAMMA,0.25
NTDELE,ALL,PRES                * Delete the specified pressures
C***                             for initial conditions
NSEL,Z
NT,ALL,PRES,200.0
NALL
EALL
LWRITE
```

```
AFWRITE
FINISH
/EXEC
/INPUT,27
FINISH
/POST1
/SHOW,TUT1,F33,1
SET,1,1                    * Plot initial conditions
PLPA,1,27,PRES
C***                      Plot At Different Times Using a Macro
*CREATE,PLOTT
SET,2,ARG1
PLPA,1,27,PRES
*END
*USE,PLOTT,20
*USE,PLOTT,50
*USE,PLOTT,100
FINISH
/DELETE,PLOTT
```

9.2 Oblique Plane Wave

```
C***   Oblique Plane Wave
/PREP7
/TITLE, PROPAGATION OF AN OBLIQUE STEP PRESSURE WAVE UNDER WATER
KAN,4                      * Tansient analysis
KAY,5,2                    * Set first and second time
C***                         derivatives of pressure to zero
ET,1,30,,1                 * Specify element type
K,1                        * Generate F.E. mesh
K,2,4
K,3,4,1
K,4,,1
K,5,,,20
K,6,4,,20
K,7,4,1,20
K,8,,1,20
L,1,2,4
L,2,3,1
L,3,4,4
L,4,1,1
V,1,2,3,4,5,6,7,8
ELSIZE,,20
VMESH,ALL
C***                      Specify Material Properties
MU,1,0                       * Zero damping
VISC,1,1440                  * Speed of sound C (m/s)
DENS,1,1000                  * Density of water (kg/m**3)
C***                      Apply Initial Conditions for the
C***                      Oblique Wave
NT,ALL,PRES,100.0
NSEL,Z,,8.1  $NRSEL,X,4    * Select nodes in Z=0 to 8 & at X=4
```

```
NT,ALL,PRES,200.0          * Specify step pressure distribution
NSEL,Z,,9.1  $NRSEL,X,3    * Select nodes in Z=0 to 9 & at X=3
NT,ALL,PRES,200.0          * Step pressure
NSEL,Z,,10.1  $NRSEL,X,2   * Select nodes in Z=0 to 10 & at X=2
NT,ALL,PRES,200.0          * Step pressure
NSEL,Z,,11.1  $NRSEL,X,1     * Select nodes in Z=0 to 11 & at X=1
NT,ALL,PRES,200.0          * Step pressure
NSEL,Z,,12.1  $NRSEL,X,0   * Select nodes in Z=0 to 12 & at x=0
NT,ALL,PRES,200.0          * Step pressure
NALL
EALL
ITER,1,,1
KBC,1
LWRITE                     * This is important for initial
C***                         conditions to take effect
TIME,0.0075                * This is the final time
ITER,50,,10                * Specifies the time step and how
C***                         often the results are written
GAMMA,0.25
NTDELE,ALL,PRES             * Delete the specified pressures
C***                          for initial conditions
NSEL,Z
NT,ALL,PRES,200.0
NALL
EALL
LWRITE
AFWRITE
FINISH
/EXEC
/INPUT,27
FINISH
/POST1
/SHOW,TUT2,F33,1
/VIEW,,,1
SET,1,1                      * Plot initial conditions
PLNS,PRES
C***                       Plot At Different Times Using a Macro
*CREATE,PLOTT
SET,2,ARG1
PLNS,PRES
*END
*USE,PLOTT,10
*USE,PLOTT,30
*USE,PLOTT,50
FINISH
/DELETE,PLOTT
```

9.3 Cylindrical Wave Propagation

```
C***   Cylindrical Wave Propagation
/PREP7
/TITLE, CYLINDRICAL WAVE PROPAGATION
KAN,4                          * Tansient analysis
KAY,5,2                        * Set first and second time
C***                             derivatives of pressure to zero
ET,1,30,,1                     * Specify element type
C***                           * Generate F.E. mesh
K,1 $,2,15 $,3,15,1 $,4,,1
L,1,2,15 $,2,3,1 $,3,4,15 $,4,1,1
A,1,2,3,4
K,5,,,15
L,1,5,15
VDRAG,1,,,,,,5
ELSIZE,,15
VMESH,ALL
C***                         Specify Material Properties
MU,1,0                         * Zero damping
VISC,1,1440                    * Speed of sound C (m/s)
DENS,1,1000                    * Density of water (kg/m**3)
C***                         Apply Initial Conditions
NSEL,X,,9.1                    * Select region for initial conditions
NUSEL,X,,7.9
NUSEL,Z,,7.1
NUSEL,Z,9.1,15
NT,ALL,PRES,200.0              * Apply pressure ahead of the wave
NINV
NT,ALL,PRES,100
NALL
EALL
GAMMA,.25                      * Damping for time integration
ITER,1,,1
KBC,1
LWRITE
C***                         Set up for Transient Solution
TIME,.003
ITER,20,,5
NTDELE,ALL,PRES                * Delete the initial conditions
NSEL,X,,9.1
NUSEL,X,,7.9
NUSEL,Z,,7.1
NUSEL,Z,9.1,15
NT,ALL,PRES,200.0              * Apply the boundary conditions
NALL
EALL
LWRITE
AFWRITE
FINISH
/EXEC
/INPUT,27
FINISH
```

```
/POST1
/SHOW,TUT3,F33,1
/VIEW,,1,1,1
/TYPE,,2
/CLABEL,,-1
SET,1,1                    * Plot initial conditions
PLNS,PRES
C***                     Plot At Different Times Using a Macro
*CREATE,PLOTT
SET,2,ARG1
PLNS,PRES
*END
*USE,PLOTT,5
*USE,PLOTT,10
*USE,PLOTT,20
FINISH
/DELETE,PLOTT
```

9.4 Room Acoustics – Harmonic Analysis

```
C***   Harmonic Analysis - Room Acoustics
/PREP7
/TITLE, ROOM ACOUSTICS-  * HARMONIC ANALYSIS
KAN,3
ET,1,30         * Acoustic elements in contact with
C***              the walls and the vibrating surface
ET,2,30,,1      * Acoustic elements in the interior (not
C***              in contact with the walls)
C***
C***              Set parameters for mesh generation
C***              ----------------------------------
XDIV=29              * Number of divisions along x-axis
YDIV=19              * Number of divisions along y-axis
ZDIV=1               * Number of divisions along z-axis
CDIV=2               * Number of divisions along radius
C***
C***                 Dimensions of the room
LEN=27
HGT=20
RAD=0.9
C***              Mesh generation
C***              ---------------
K,1
K,2,LEN
K,3,LEN,HGT
K,4,,HGT
K,5,,,-1
K,6,LEN,,-1
K,7,LEN,HGT,-1
K,8,,HGT,-1
L,1,5,1
L,2,6,1
```

```
L,3,7,1
L,4,8,1
CIRC,3,RAD,7,2,90,2
ADRAG,5,6,,,,, 3
PIO4=ATAN(1)
LENC=COS(PIO4)
LENC=LENC*RAD
HGTC=HGT-LENC
LENC=LEN-LENC
K,15,,HGTC
K,16,,HGTC,-1
K,17,LENC
K,18,LENC,,-1
L,1 ,17,XDIV
L,10,15,XDIV
L,11,4 ,XDIV
L,17,10,YDIV
L,15, 1,YDIV
L,2 ,9 ,YDIV
L,5 ,18,XDIV
L,13,16,XDIV
L,14,8 ,XDIV
L,18,13,YDIV
L,16,5 ,YDIV
L,6 ,12,YDIV
ELSIZE,,CDIV
V,1 ,17,10,15, 5,18,13,16
V,15,10,11,4 ,16,13,14,8
V,17,2 ,9 ,10,18,6 ,12,13
VMESH,ALL
C***
C***              Material properties
C***              -------------------
DENS,1,2.35E-3       * (Rho) density of air. (lb-sec**2)/ft
VISC,1,1100.0        * (C) speed of sound in air. ft/sec
MU,1,0.04            * (Beta) admittance of the walls
C***
DENS,2,2.35E-3
VISC,2,1100.0
MU,2,0.70            * (Beta) admittance of the damping
C***                   Material
DENS,3,2.35E-3
C***
VISC,3,1100.0
MU,3,0.0             * (Beta) admittance is set to zero for
C***                   Interior elements
NSEL,Y,0.0
NRSEL,X,12,15
ENODE
MAT,2
EMOD,ALL             * Elements which have the damping
C***                   Materail at the interface
C***
```

```
C***              Boundary conditions
C***              -------------------
EALL
NSEL ,X,0.0
NASEL,Y,0.0
NASEL,X,LEN
NASEL,Y,HGT
D,ALL,UX,,,,,UY,UZ   * Constrain all displacements to zero
C***                   At the walls
LOCAL,11,1,27,20
NASEL,X,RAD
ENODE
EINV
TYPE,2
MAT,3
EMODIF,ALL            * Interior elements are specified as
C***                   Type=2 and material=3
EALL
NSEL,X,RAD
PSF,ALL,,,1.0         * Specify the vibrating surface as
C***                   Fluid-structure interface
NROTAT,ALL
D,ALL,UX,.01          * Radial vibration amplitue of the
C***                   Vibrating surface
D,ALL,UY,,,,,UZ
CSYS,0
NSEL ,X
NASEL,X,LEN
NASEL,Y
NASEL,Y,HGT
PSF,ALL,,,1.0         * Specify the walls as fluid-structure
C***                   Interface
HARF,80,80            * Frequency of excitation = 80 Hz
NALL
EALL
ITER,1,,1
AFWRITE
FINISH
/EXEC
/INPUT,27
FINISH
C***              Plot the Standing Wave Pattern (f = 80 Hz)
C***              ------------------------------------------
/POST1
/SHOW,TUT4,F33,1
SET,1,1
/EDGE,,1
PLNS,PRES
FINISH
```

9.5 Room Acoustics – Modal Analysis

```
C***   Modal Analysis - Room Acoustics
/PREP7
/TITLE, ROOM ACOUSTICS-  * MODAL ANALYSIS
KAN,2
KAY,1,-1       * Full Subspace Iterations to Extract Eigenvalues
KAY,2,10
KAY,3,0
KAY,7,15
ET,1,30,,1     * Acoustic Element Without F/S Interface
C***
C***             Set Parameters for Mesh Generation
C***             ---------------------------------
XDIV=10             * Number of divisions along X-axis
YDIV=5              * Number of divisions along Y-axis
ZDIV=1              * Number of divisions along Z-axis
C***
C***             Mesh Generation
C***             ---------------
K,1
K,2,27
K,3,27,20
K,4,,20
K,5,,,-1
K,6,27,,-1
K,7,27,20,-1
K,8,,20,-1
L,1,5,ZDIV
L,2,6,ZDIV
L,3,7,ZDIV
L,4,8,ZDIV
L,1,2,XDIV
L,3,4,XDIV
L,2,3,YDIV
L,4,1,YDIV
L,5,6,XDIV
L,7,8,XDIV
L,6,7,YDIV
L,8,5,YDIV
V,1,2,3,4,5,6,7,8
VMESH,ALL
C***             Material Properties
C***             -------------------
DENS,1,2.35E-3       * (RHO) Density of air. (lb-s**2)/ft
VISC,1,1100          * (C) Speed of sound in air. ft/s
MU,1,0.0             * (BETA) Admittance is set to zero
C***                   for modal analysis
C***
C***             Couple the Nodes at Z=0 and Z=-1
C***             planes for the 2-D problem
C***             --------------------------------
ICP=0
```

```
*CREATE,MAC
NSEL,Y,LY
ICP=ICP+1
LX=LX+2.7
NRSEL,X,LX
CP,ICP,PRES,ALL
*END
LX=-2.7  $LY=4
*USE,MAC
RP11
LX=-2.7  $LY=8
*USE,MAC
RP11
LX=-2.7  $LY=12
*USE,MAC
RP11
LX=-2.7  $LY=16
*USE,MAC
RP11
LX=-2.7  $LY=0
*USE,MAC
RP11
LX=-2.7  $LY=20
*USE,MAC
RP11
NSEL,Z
NALL
ITER,1,1,1
AFWRITE
FINI
/INPUT,27
FINI
/DELETE,MAC
```

9.6 Impedance Tube Problem – Harmonic and Modal Analysis

```
C***   Effect of Element Size on the Solution
/PREP7
/TITLE, IMPEDENCE TUBE PROBLEM - HARMONIC & MODAL ANALYSES
KAN,2                          * Modal analysis
ET,1,30,,1                     * Only pressure d.o.f. chosen
C***                         Define Material Properties
DENS,1,1.12                    * Air
VISC,1,344.424                 * Air
C***                         Generate the F.E. Mesh
C***                         (Parametrize for Varying Element Size)
NDV=7
SIZ=(.5334/NDV)
N1=NDV+1
N2=N1*5
N,1
N,N1,.5334
```

```
FILL,1,N1
NGEN,5,N1,1,N1,1,,.0508
NGEN,4,N2,1,N2,1,,,.0254
L1=N1+2
L2=N1+1
L3=N2+1
L4=N2+2
L5=N1+N2
L5=L5+2
L6=L5-1
E,1,2,L1,L2,L3,L4,L5,L6
EGEN,4,N1,-1
EGEN,3,N2,-4
EGEN,NDV,1,-12
C***                          Couple Nodes at Each X-plane(1-D prob.)
CPSIZE,20
DX=0.0
DN=1
ICP=2
*CREATE,MAC1
NSEL,X,DX
CP,ICP,PRES,ALL
M,DN,PRES
ICP=ICP+1
DX=DX+SIZ
DN=DN+1
*END
*USE,MAC1
RP8                              * For different NDV change 8 to NDV+1
NALL
EALL
C***
ITER,1,1,1
KBC,1
AFWRITE
FINI
/INPUT,27
FINI
C***
/PREP7
RESUME
/TITLE, IMPEDENCE TUBE PROBLEM - HARMONIC & MODAL ANALYSES
KAN,3                            * Harmonic analysis
ET,2,30                          * Pres. and displacement d.o.f's
NSEL,X,0
ENODE
TYPE,2
MAT,1
EMOD,ALL
PSF,ALL,,,1.0                    * Specify the interface
D,ALL,UY,,,,,UZ                  * Constrain UY & UZ for the 1-D prob.
C***                            Couple Nodes at Each X-plane(1-D prob.)
CPSIZE,20
```

```
CP,1,UX,ALL
M,1,UX
NALL
EALL
KBC,1
ITER,1,,1
C***                    Set up for Harmonic Response at
C***                    Different Frequencies.
FQN=0
VEL=1.27                   * Piston velocity
*CREATE,MAC2
FQN=FQN+100
C***                       * Equivalent displacement for the
C***                         piston velocity
VAL=VEL/FQN
VAL=VAL/6.2831853
D,1,UX,,-VAL
HARF,FQN,FQN
LWRITE
*END
*USE,MAC2
RP10
AFWRITE
FINI
/INPUT,27
FINI
/POST26
C***                    Display PRES at Corner Nodes on Piston
M1=1
M2=N2-NDV
M3=N2*3
M3=M3+1
M4=N2*4
M4=M4-NDV
DISP,2,M1,PRES
DISP,3,M2,PRES
DISP,4,M3,PRES
DISP,5,M4,PRES
PRVAR,1,2,3,4,5
FINI
/DELETE,MAC1
/DELETE,MAC2
```

Appendix A

List of all Files on FLOPPY DISK from SWANSON Co.

A.1 Coupled Field Analysis

FILE	TUTORIAL	SUBJECT DN-T021 1.5.1989	STIF
EXMPL1	5.3-5.12	TOKAMAK FUSION REACTOR COIL ANALYSIS	5
EXMPL2	5.15-5.20	HOT DIE FORGING THERMAL-STRESS ANALYSIS	13-34
EXMPL3	5.23-5.27	SLOT EMBEDDED CONDUCTOR	13
EXMPL4	5.32-5.34	DOUBLE-MASS LOADED TRANSDUCER	5

A.2 Revision 4.4 Changes Tutorial

FILE	TUTORIAL	SUBJECT DN-T023 1.5.1989	STIF
EXMPL1	5.2	TIME STEP OPTIMIZATION	42
EXMPL2	5.3	SPIN SOFTENING	45
EXMPL3	6.1	EXAMPLE TO COMPUTE HEAT LOSS THROUGH SIDES	55
EXMPL4	8.7	SLOT EMBEDDED CONDUCTOR	13
EXMPL5	8.8	AXISYMMETRIC FLOW-FLOW BETN. COAXIAL CYLINDERS	15
EXMPL6	8.8	THERMAL EFFECTS IN HAGEN-POISSEULLE FLOW	15
EXMPL7	8.15	TEST STIF43 ELASTICALLY	63-45-43
EXMPL8	8.17	OPTIMIZATION OF FIBER ANGLE OF COMPOSITE MATERIAL	46
EXMPL9	8.10	TEST STIF26 WITH THREE NODES AND LARGE ROTATIONS	26
EXMPL10	3.3	CURVED PLATE WITH HOLE	63

A.3 Postprocessing

FILE	TUTORIAL	SUBJECT DN-T018 1.5.1989	STIF
EXMPL1	1.1-1.2	PIPE WITH COOLING FINS	55 70
EXMPL2	3.8-3.9	VECTOR AND RASTER CONTOUR PLOTS	42
EXMPL3	3.18-3.19	ANIMATION OF CANTILEVER BEAM	42
EXMPL4	3.20-3.32	PATH OPERATIONS	42
EXMPL5	3.36-3.42	VECTOR DATA	70
EXMPL6	3.44-3.45	LINE ELEMENT PROCESSING	3 12
EXMPL7	3.54-3.60	DUMP TRUCK - MODAL ANALYSIS	63
EXMPL8	3.62-3.66	INTERFACING BEAMS - STATIC ANALYSIS	3 12
EXMPL9	3.68-3.74	CANTILEVER BEAM WITH DISTRIBUTED LOAD	45
EXMPL10	3.75-3.81	SYMMETRIC ANALYSIS - CANTILEVER BEAM	45
EXMPL11	3.84-3.86	ERROR NORM EXAMPLE - ELLIPTIC MEMBRANE	42
EXMPL12	3.89-3.95	ADAPTIVE MESHING OF ELLIPTIC MEMBRANE	2

A.4 Nonlinear Dynamics

FILE	TUTORIAL	SUBJECT DN-T016 1.5.1989	STIF
EXMPL1	5.1	NONLINEAR IMPULSIVELY LOADED CLAMPED PLATE	43
EXMPL2	5.2	PIPE WHIP ANALYSIS INCLUDING RESTRAINT	20
EXMPL3	5.3	LARGE DISPLACEMENT ELASTIC DYN. RESP. OF CLAMPED BEAM	82
EXMPL4	5.4	IMPULSIVELY LOADED SIMPLY SUPPORTED PLASTIC BEAM	82
EXMPL5	5.5	SLIDER-CRANK MECHANISM UNDER IMPACT	3-21-40

A.5 The Four – Node Plastic Shell Element STIF 43

FILE	TUTORIAL	SUBJECT DN-T017 1.5.1989	STIF
EXMPL1	5.2	PATCH TEST	43
EXMPL2	5.4-5.7	ELEM.PERFORMANCE FOR THICK TO MODERATELY THIN CASES	43
	A.2-A.4	SIMPLY SUPPORTED WITH UNIFORM LOAD	43
EXMPL3	5.4-5.7	ELEMENT PERFORMANCE-FIXED WITH UNIFORM LOAD	43
	A.4.A.6		43
EXMPL4	5.4-5.7	ELEMENT PERFORMANCE-SIMPLY SUPPORTED,DISTORTED	43
EXMPL5	5.4-5.7	ELEMENT PERFORMANCE-FIXED,DISTORTED	43
EXMPL6	5.8	THICK AND THIN CANTILEVER BEAM	43
EXMPL7	5.10	LINEAR ANALYSIS OF A CYLINDRICAL(SCORDELIS-LO) SHELL	43
EXMPL8	5.12	LINEAR ANALYSIS OF A PINCHED CYLINDER	43
EXMPL9	5.14	FREE VIBRATION OF SIMPLY SUPPORTED RECTANGULAR PLATE	43
EXMPL10	5.16	VIBRATION/PRESTRESSED SIMPLY SUPPORTED RECT. PLATE	43
EXMPL11	5.18	VIBRATORY BEHAVIOR OF TWISTED CANTILEVER PLATES	43
EXMPL12	5.25	TWISTED PLATE VIBRATION UNDER CENTRIFUGAL LOADING	43
EXMPL13	5.28	BUCKLING OF A SEMICIRCULAR ARCH	43
EXMPL14	5.30	BUCKLING OF A CLOSED RING	43
EXMPL15	5.32	BUCKLING CIRCULAR CYLINDRICAL SHELL-BOTH ENDS HINGED	43
EXMPL16	5.35	LARGE DEFLECTION ANALYSIS OF A CLAMPED SQUARE PLATE	43
EXMPL17	5.37	SIMPLY SUPPORTED SQUARE PLATE-UNIFORM IMPULSE LOAD	43
EXMPL18	5.39	IMPULSIVELY LOADED CLAMPED BEAM	43
EXMPL19	5.41	LARGE DISPLAC. RESPONSE OF A SHALLOW SPHERICAL SHELL	43
EXMPL20	5.43	NONLINEAR ANALYSIS OF A BARREL VAULT ROOF	43

A.6 Solid Modeling

FILE	TUTORIAL	SUBJECT DN-T022 1.5.1989	STIF
EXMPL1	6.1	SPUR GEAR CONTACT ANALYSIS (MODEL 1)	82
EXMPL2	6.7	MODEL 2 (DRIVEN GEAR)	1 82
EXMPL3	6.14	COMBINE MODELS 1 AND 2	
EXMPL4	6.16	GENERATION PASS	0 1
EXMPL5	6.20	USE PASS	50 52
EXMPL6	6.23	STRESS PASS	
EXMPL7	6.23	POSTPROCESSING	
GTOOTH		MACRO FILE FOR USE WITH THE FIRST 7 EXAMPLES	
EXMPL8	6.25	OPTIMIZATION OF A FILLET RADIUS	83
EXMPL9	6.34	TURBINE BLADE 3-D MODELING (not in this note !)	

EXMPL10	1.3	PLASTIC BLOWER WHEEL HOUSING	93
EXMPL11	2.10	RETAINING RING MODEL	2
EXMPL12	2.17	20 NODE BRICKS WITH 10 NODE TETRAHEDRA (TOP MOD)	29 95
EXMPL13	2.17	8 NODED BRICKS WITH 20 NODED TETRAHEDRA (BOT MOD)	45
EXMPL14	4.6	VOLUME GENERATION WITH CONTINUOUS MESH	2
EXMPL15	4.14	2-DIMENSIONAL CONTACT EXAMPLE	0 1 12
EXMPL16	4.17	3-DIMENSIONAL CONTACT EXAMPLE	0 12 63 92
EXMPL17	4.22	CONNECTING DISSIMILAR MESHES WITH CONSTRAINT EQ'S	
PRIM.USR		PRIMITIVE GEOMETRY USER FILE	

A.7 Composite Structures

FILE	TUTORIAL	SUBJECT	DN-T020	1.5.1989	STIF
EXMPL1	5.1	CLAMPED PLATE ANALYSIS OF UNSYMMETRIC LAMINATE			99
EXMPL2	5.15	INTERLAMINAR STRESSES IN A CANTILEVER BEAM			64 99
EXMPL3	5.30	FAILURE OPTIMIZATION OF A THICK-WALLED PRESSURE VESSEL UNDER INTERNAL PRESSURE.			46
EXMPL4	5.46	BUCKLING ANALYSIS OF A CYLINDRICAL COMPOSITE PANEL WITHA HOLE			91
DOCU1		DOCUMENTATION FOR MACRO FILE LAYUP			
LAYUP		MACRO FILE -- GENERATES REAL CONSTANTS REQUIRED FOR COMPOSITE ELEMENTS STIF46 AND STIF99			
DOCU2		DOCUMENTATION FOR MACRO FILE COLIST			
COLIST		MACRO FILE--LISTS ALL PERTINENT DATA FOR EVERY COMPOSITE			

A.8 Fracture Mechanics using the ANSYS Program

FILE	TUTORIAL	SUBJECT	DN-T020	1.5.1989	STIF
EXMPL1	4.8/9	SINGLE-EDGE CRACK ; ELASTIC ANALYSIS SHOWS KCALC USE AND J-INTEGRAL EVALUATIONS FOR KI			2
EXMPL2	4.10	SINGLE-EDGE CRACK;SHOW NODAL FORCES AND DISPLACEMENTS USED IN THE HAND CALCULATIONS FOR VIRTUAL CRACK CLOSURE METHOD OF KI.			2
EXMPL3	4.11/12	SINGLE-EDGE CRACK; DEMONSTRATE POST1 COMPUTATION OF ENERGY DIFFERENCE USED IN VIRTUAL CRACK EXTENSION TECHNIQUE FOR KI COMPUTATIONS. (COARSE MODEL)			2
EXMPL4	4.18/19	ELASTIC ANALYSIS OF INCLINED EDGE CRACK			2
EXMPL5	4.32/34	ANALYSIS OF COMPACT TENSION SPECIMEN (FINE MODEL) INCLUDES POST1 J-INTEGRAL PATH EVALUATIONS.			2
EXMPL6	4.34/35	ANALYSIS OF COMPACT TENSION SPECIMEN (COARSE MODEL) INCLUDES POST1 J-INTEGRAL PATH EVALUATIONS.			2
EXMPL7	4.36/37	ANALYSIS OF COMPACT TENSION SPECIMEN (COARSE MODEL) USING POST26 ESTIMATE OF J BASED ON THE LOAD VS. LOAD-POINT DISPLACEMENT CURVE.			2
EXMPL8	4.45/48	ELASTIC ANALYSIS OF 3-D SEMI-CIRCULAR SURFACE FLAW			92
JIN1		MACRO FILE USED WITH EXAMPLES 5 AND 6.			
JIN2		MACRO FILE USED WITH EXAMPLE 1.			

A.9 Acoustic Pressure Wave Propagation

FILE	TUTORIAL	SUBJECT DN-T015 1.5.1989	STIF
EXMPL1	A.1	1-D PLANE WAVE	30
EXMPL2	A.3	OBLIQUE PLANE WAVE	30
EXMPL3	A.5	CYLINDRICAL WAVE PROPAGATION	30
EXMPL4	A.7	ROOM ACOUSTICS - HARMONIC ANALYSIS	30
EXMPL5	A.10	ROOM ACOUSTICS - MODAL ANALYSIS	30
EXMPL6	A.12	IMPEDANCE TUBE PROBLEM - HARMONIC AND MODAL ANALYSES	30